

Plastic Injection Molding

...material selection
and product design
fundamentals

By Douglas M. Bryce

Volume II: *Fundamentals of
Injection Molding* series



Published by the
Society of Manufacturing Engineers
Dearborn, Michigan

Copyright © 1997 by Douglas M. Bryce and Society of Manufacturing Engineers

987654321

All rights reserved, including those of translation. This book, or parts thereof, may not be reproduced in any form or by any means, including photocopying, recording, or microfilming, or by any information storage and retrieval system, without permission in writing of the copyright owners.

No liability is assumed by the publisher with respect to the use of information contained herein. While every precaution has been taken in the preparation of this book, the publisher assumes no responsibility for errors or omissions. Publication of any data in this book does not constitute a recommendation or endorsement of any patent, proprietary right, or product that may be involved.

Library of Congress Catalog Card Number: 97-068807

International Standard Book Number: 0-87263-488-4

Additional copies may be obtained by contacting:

Society of Manufacturing Engineers
Customer Service
One SME Drive
Dearborn, Michigan 48121
1-800-733-4763

SME staff who participated in producing this book:

Donald A. Peterson, Senior Editor
Rosemary K. Csizmadia, Production Team Leader
Dorothy M. Wylo, Production Assistant
Jennifer L. Courter, Editorial Assistant
Karen M. Wilhelm, Manager, Reference Publications
Cover design by Judy D. Munro, Manager, Graphic Services

Preface

This is the second volume in the Society of Manufacturing Engineers' *Fundamentals of Injection Molding* series intended for those requiring an awareness of the basics before being exposed to the trials and tribulations that accompany typical seat-of-the-pants accumulation of knowledge. It is also intended as a refresher (and maybe a source of new data) for those who have been in the business for awhile but wish to stay on top of things. Although the majority of information herein addresses the material selection and product design phases of the thermo-plastic injection molding process, some material from Volume 1 is included to provide the continuity necessary to adequately describe the two phases within the context of the injection molding process as a whole. The two phases are addressed in the same book because it is impractical to have one without the other. A good design requires the selection of the proper material.

My decision to publish a source of basic, fundamental information on injection molding was based on my belief that those who are interested in getting an initial grasp of the subject and also wish to have a reference tool of charts, diagrams, and data that can be used for years to come, need an easily understood source for this information. I believe this goal is accomplished in the publication of this series. But it could not have been done without the cooperation and involvement of many people. I wish to take this opportunity to thank the following for their contributions to the achievement of this goal.

- *Society of Manufacturing Engineers*, especially Don Peterson, for having the patience and fortitude to persevere through the publishing process.
- *AEC, Incorporated*, for the use of photographs and information concerning various pieces of equipment manufactured by them.
- *Branson Ultrasonics, Incorporated*, for the use of photographs and information concerning various pieces of equipment manufactured by them.
- *GE Plastics*, for the use of photographs and information concerning various plastics manufactured by them.
- *Hoechst-Celanese*, for their assistance and permission to use information and data concerning various plastics manufactured by them.
- *Lexmark, International*, especially Steve Spanoudis, for permission to use chart and table data concerning various acronyms and tradenames.
- *Perkin-Elmer Corporation*, for the use of photographs and information concerning various pieces of equipment manufactured by them.
- *Texas Plastic Technologies*, for allowing the use of chart, table, research and archive data concerning various plastic materials, processes, tests, and programs.

- *United Silicone, Incorporated*, for the use of photographs and information concerning various pieces of equipment manufactured by them.
- *Van Dorn and Cincinnati-Milacron*, for use of photographs of various equipment they manufacture.

Please accept this book as it is intended. I hope your experiences will reward you with much success and few failures. Welcome to the world of plastics.

I would like to dedicate this book to the memory of my mother, who passed away suddenly during its preparation stages. She was a positive influence in my life and I (along with everyone else who knew her) will miss her very much.

Douglas M. Bryce
Georgetown, Texas 1997

Table of Contents

<i>Preface</i>	xxiii
----------------------	-------

Chapter 1 – Understanding the Injection Molding Process

Evolution of the Process	1
Charting Industry Evolution	1
Evolution of Screw Concept and Evaluation of Plunger	2
The Process	3
Categorizing the Parameters	4
Temperature/Pressure/Time/Distance	
Horizontal versus Vertical Molding	20
Advantages of Horizontal Molding	20
Advantages of Vertical Molding	21
The Need for Process Controls	22
Part Quality	22
Part Cost	23
Parameter Effects	23
The Setup Sheet	24
Controlling Shrinkage	26
The Effects of Temperature Adjustments	29
The Effects of Pressure Adjustments	30
Post-mold Shrinkage	30
Minimizing Molded-in Stress	32
Summary	33
Questions	34

Chapter 2 – The Language of Plastics

Definitions Relating to Materials	37
The Definition of Plastic	37
Polymerization	37
Basic Molecular Structure/Copolymers/Alloys and Blends	
Thermoplastics versus Thermosets	42
Amorphous versus Crystalline	43
Amorphous Materials/Crystalline Materials/Comparison of Amorphous and Crystalline Molecular Chains	
Elastomers	44

How Plastics are Made	45
Molecular Weight and Distribution	46
Influence of Time and Temperature	46
Summary	47
Questions	48
 <i>Chapter 3 – Reinforcements and Fillers</i>	
Why Reinforcements and Fillers are Used	49
What Are They?	49
Reinforcements	49
Fiberglass/Organic Fibers/Boron/Carbon-graphite/Ceramics (Metal Oxides)/Metals	
Fillers	53
Colorants/Plasticizers/Heat Stabilizers/Antioxidants/Flame Retardants/Ultraviolet (UV) Absorbers/Antistatic Agents/ Blowing Agents/Lubricants	
Summary	57
Questions	58
 <i>Chapter 4 – Comparing Amorphous and Crystalline Materials</i>	
Identifying the Differences	59
Molecular Structure	59
Crystalline Polymers/Amorphous Polymers/ Crystalline/Amorphous Polymers	
Process Temperature Requirements (injection unit)	61
Process Temperature Requirements (mold)	63
Crystalline/Amorphous/General Differences Based on Crystallization	
Liquid Crystalline Materials	66
Summary	67
Questions	67
 <i>Chapter 5 – Basic Material Properties</i>	
Property Descriptions and Terminology	69
Physical Properties	69
Density/Specific Gravity/Shrinkage Rate/Water Absorption/ Transparency (Opacity)/Modulus (Toughness)/Elasticity/ Plasticity/Ductility/Brittleness/Notch Sensitivity/Lubricity/ Isotropy and Anisotropy	

Mechanical Properties	74
Stress and Strain/Modulus of Elasticity/Poisson's Ratio/Shear	
Stress and Shear Strain/Shear Modulus/Direct Shear/	
Compressive Strength and Modulus/Flexural Strength and	
Modulus/Creep/Impact Strength/Fatigue Endurance	
Thermal Properties	83
Melting Point/Glass Transition Temperature/Melt Index/	
Vicat Softening Point/Heat Deflection (Distortion)	
Temperature/Coefficient of Linear Thermal Expansion/	
Thermal Conductivity/Temperature Index/Flammability	
Electrical Properties	85
Volume Resistivity/Surface Resistivity/Dielectric Strength/	
Dielectric Constant (Permittivity)/Arc Resistance/Comparative	
Tracking Index (CTI)	
Environmental Concerns	88
Stress/Exposure Factors/Temperature/Chemicals/Weathering	
Summary	92
Questions	93

Chapter 6 – Applications and Properties of Specific Materials

Characteristics and Applications	95
Common Thermoplastics	95
Acrylonitrile-butadiene-styrene (ABS)/Acetal/Acrylic/Acrylic-	
styrene-acrylonitrile (ASA)/Cellulose/Fluoroplastics/Ionomers/	
Liquid Crystal Polymers/Methyl Pentenes/Nylon (Polyamide)/	
Polyallomers/Polycarbonates/Polyetheretherketone (PEEK)/	
Polyethylene/Polyimides/Polyphenylene Oxide (PPO)/	
Polyphenylene Sulfide (PPS)/Polypropylene/Polystyrene/	
Polysulfones/Polyvinylchloride (PVC)/Thermoplastic Polyesters	
(PBT, PET)/Thermoplastic Copolyesters (PCTA, PCTG, PETG)/	
Polyarylate (Wholly Aromatic Polyester [ARP])/Thermoplastic	
Rubbers (TPR)	
Elastomers	103
Natural Rubber (NR)/Isoprene Rubber (IR)/Styrene-butadiene	
Rubber (SBR)/Neoprene (CR)/Nitrile Rubber (NBR)/Butyl	
Rubber (IIR)/Ethylene Rubber (EPM)/Ethylene Terpolymer Rubber	
(EDPM)/Hypalon™ (CSM)/Acrylic Rubber (ABR)/Polysulfide	
Rubber (T)/Silicones (SI, FSI, PSI, VSI, PVSI)/Urethanes (U)/	
Fluoroelastomers (FPM)/Butadiene Rubbers (BR)/Carboxylic	
(COX)/Epichlorohydrin Rubbers (CO, ECO)	
Star Diagrams	106

Structural Foams	106
Summary	107
Questions	107

Chapter 7 – Determining Injection Molding Costs

Is It Worth It?	135
What Information is Needed?	135
Material Costs	136
Use of Regrind/Impact of Runner versus Shot Size	
Labor Costs	139
Machine Costs	139
Determining Machine Size/Determining Machine Location/ Calculating Molding Process Costs/Should Operator Cost be Included?/Estimating Setup Charges	
Tooling Costs	144
Standard Practice/Amortizing Tool Costs/Maintenance Costs	
Adding It All Up	146
Secondary Operations (Including Packaging)	146
Summary	147
Questions	148

Chapter 8 – Secondary Operations

Defining Secondary Operations	149
When to Consider Secondary Operations	150
When Volumes are Small	150
When Tooling Costs are Excessive	150
When Time to Build the Mold Jeopardizes Marketing Schedules	150
When a Labor-heavy Environment Exists	151
Assembly Operations	151
Ultrasonic Welding	151
Energy Directors/Ease of Welding/Variables that Influence Weldability	
Hot Gas Welding	157
Induction (Electromagnetic) Bonding	158
Spin Welding (Friction Welding)	159
Adhesive Bonding	159
Machining Operations	160
Drilling and Tapping Thermoplastics	160
Drilling and Tapping Thermosets	164

Reaming Thermoplastics and Thermosets	164
Turning and Milling Thermoplastics and Thermosets	164
Automated Shape Cutting	165
Water Jet	165
Laser	166
Surface Finishes and Decorating Procedures	166
Preparation of Surface	166
Flame Treatment/Corona Discharge/Plasma Process/ Acid Etch	
Applied Finishes	167
Painting/Plating (Electroplating)/Vacuum Metallizing (Deposition)/Hot Stamping/Pad Printing (Heat Transfer)/ Screen Printing	
In-process Finishes	173
Molded-in Color/Molded-in Symbols/Two-color (Two-shot) Molding/Textured Surface/In-mold Overlays	
Summary	177
Questions	177

Chapter 9 – Testing and Failure Analysis

Distinguishing the Terms	179
Testing	179
Electrical Testing	179
Conditioning Samples/Dielectric Strength/Dielectric Constant/ Volume Resistivity/Surface Resistivity/Arc Resistance	
Physical Testing	181
Shrinkage Rate/Density/Water Absorption/Moisture Content/ Melt Flow Index	
Mechanical Testing	186
Tensile Strength/Compressive Strength/Flexural Strength/ Creep/Impact	
Thermal Testing	189
Melting Point/Heat Deflection Temperature/Vicat Softening Temperature/Flammability/Limiting Oxygen Index	
Failure Analysis	195
As a Design Tool	195
Stress	195
Differential Scanning Calorimeter	196
Stress/Moisture/ T_m and T_g Points/Regrind Percentage/ Crystallinity	

Calculating Glass Content	199
Furnace Method/TGA Method	
Summary	201
Questions	202
 <i>Chapter 10 – The Product Design Session</i>	
Brainstorming	203
Who Are the Players?	203
What Takes Place?	204
What Does the Product Do?/How Will the Product be Made?/ When Should Production Start?/Where Will It be Built?/How Much Should It Cost?	
Project Chart Samples	209
Product Development/Project Management Process/Process Management/Product Development Time Line	
Why Projects Fail	216
Uninvolved Users/Undisciplined Management System/ Ineffective Communication/Uncontrolled Changes/ Inappropriate Technology/Poorly Defined Products	
Summary	220
Questions	220
 <i>Chapter 11 – Determining Product Requirements</i>	
Defining the Product	223
Typical Project Example	224
Phase 0	225
Phase 1	225
Phase 2	225
Phases 3 through 8	226
Determining Actual Requirements	227
Functional Aspects.....	227
Color/Fit/Appearance	
Physical and Mechanical Aspects	231
Mechanical Assembly Methods/Other Assembly Methods	
Thermal Aspects	248
Electrical Aspects	249
Flammability Considerations	249
Weather and Exposure Limits	250
Safety and Environmental Issues	251
Summary	252
Questions	253

Chapter 12 – Product Design Considerations

Design Guidelines	255
Wall Thickness Recommendations	255
Uniform Wall Thickness Requirement	257
Knit Lines	258
Sharp Corners	259
Sink Marks	261
Warpage and Bowing	262
Draft	264
Ejector Pin Marks and Gate Location Restrictions	269
Gate Marks	270
Dimensions and Drawing Notes	271
What the Vendor Needs	272
Common Design Considerations	276
Product Design Checklist	279
Summary	281
Questions	283

Chapter 13 – Miscellaneous Determinations

Annual Volumes	285
Material Handling Requirements	286
Secondary Operations	286
Packaging Requirements	287
Testing Requirements	288
Summary	289
Questions	290

Chapter 14 – Selecting a Vendor

Overview	293
List of Requirements	293
Dedicated Specialty or Full Service?	293
Dedicated Specialty/Full Service	
What to Look For	295
Basic Investigation	296
Note Condition of Plant and Equipment/Ask to See Product Samples/Check Delivery Performance/Check Quality Rating/ Investigate Specialty Areas/Determine Equipment Availability/ Examine Bidding Practices	
Familiarity	297
Normal Tolerance Levels/Materials/Molding Processes/ Equipment Type, Age, and Size/Subcontracting	

Capabilities	298
Volumes/Tool Design, Build, and Repair/Computerized Operations/Statistical Process Control (SPC) and Other Quality Programs	
Responsibilities	299
Qualification Form	300
Summary	304
Questions	304

Chapter 15 – Prototype Overview

The Value of Prototyping	307
Form	307
Fit	308
Function	308
Appearance	308
The Prototype Option	309
Typical Methods and Recommended Quantities	309
Fabrication	309
Thermoforming	309
Casting	310
Molding	310
Virtual Prototyping	310
Rapid Prototyping (RP)	310
Stereolithography Apparatus (SLA)/Selective Laser Sintering (SLS)/Laminated Object Manufacturing (LOM)/Ballistic Particle Manufacturing (BPM)/Solid Ground Curing (SGC)	
Which is Best?	312
Summary	313
Questions	313

Chapter 16 – A Primer on Molds

Description of the Mold	315
The "A" and "B" Plates	315
The Cavity Image	315
The Injection Half of the Mold	316
The Sprue Bushing	317
Runners	317
Flash	318
The Ejector Half of the Mold	318

Ejector Pins	318
Ejector Plates	319
Knockout Rod	319
Mold Design Basics	320
Gate Location	320
Runner Cross-section	321
Venting Concepts	321
Typical Tool Life	324
Ownership of the Mold (and Other Tools)	326
What to Expect from Molds	326
Summary	328
Questions	329

Chapter 17 – Understanding Defects

Overview	331
What Causes Defects?	332
Common Defects and Remedies	333
Black Specks or Streaks	333
Excessive Residence Time in Barrel/Contaminated Raw Material	
Blisters	334
Back Pressure Too Low	
Blush	334
Mold Temperature Too Low	
Bowing	334
Clamp Opens Too Quickly/Improper Handling	
Brittleness	335
Excessive Moisture	
Bubbles (Voids)	335
Section Thickness Too Great/Excessive Moisture	
Burn Marks	335
Improper Venting/Excessive Regrind	
Clear Spots	336
Barrel Temperature Too Low/Inconsistent Cycles	
Cloudy Appearance	336
Uneven Packing of Cavity	
Contamination	337
Improper Regrind/Poor Housekeeping	
Cracking	337
Molded-in Stresses/Insufficient Draft or Polish	

Crazing	338
Delamination	338
Foreign Materials or Additives/Excessive Mold Release	
Discoloration	338
Improper Mold Temperature/Contamination	
Flash	339
Excessive Injection Pressure/Improper Flow Rate	
Flow Lines	339
Inadequate Injection Pressure	
Gloss (Low)	339
Inadequate Injection Pressure/Mold Temperature Too Low	
Jetting	340
Improper Gate Design or Location	
Knit Lines (Weld Lines)	340
Barrel Temperature Too Low	
Nonfill (Short Shots)	340
Insufficient Material Feed/Insufficient Venting	
Shrinkage (Excessive)	341
Mold Temperature Too High/Early Gate Opening	
Sink Marks	341
Excessive Rib Thickness	
Splay (Silver Streaking)	342
Excessive Moisture	
Warpage	342
Inadequate Injection Pressure or Time	
Troubleshooting Tips	342
Summary	343
Questions	343
<i>Appendix A – Material Tradenames and Descriptions</i>	<i>345</i>
<i>Appendix B – Common Material Acronyms</i> <i>and Their Meanings</i>	<i>355</i>
<i>Appendix C – Material Rankings for Various Properties</i>	<i>359</i>
<i>Answers to Chapter Questions</i>	<i>361</i>
<i>Index.....</i>	<i>371</i>

Understanding the Injection Molding Process 1

EVOLUTION OF THE PROCESS

In 1868, a gentleman by the name of John Wesley Hyatt developed a plastic material called *celluloid* and entered it in a contest created by a billiard ball manufacturer. The purpose of the contest was to find a substitute for ivory, which was becoming expensive and difficult to obtain. Celluloid was actually invented in 1851 by Alexander Parkes, but Hyatt perfected it to where it could be processed into a finished form. He used it to replace the billiard ball ivory and won the contest's grand prize of \$10,000, a rich sum in those days. Unfortunately, after the prize was won, some of the celluloid billiard balls exploded on impact during a demonstration (due to the instability and high flammability of the material) and further refinement was required to use it in commercial ventures. Nonetheless, the plastics industry was born, and it would begin to flourish when John Wesley Hyatt and his brother Isaiah patented the first injection molding machine in 1872. They used this machine to injection mold celluloid plastic. Over the next 40 to 50 years others began to investigate this new process and expand its application to manufacturing such items as collar stays, buttons, and hair combs. By 1920, the injection molding industry was well entrenched, and it has been booming ever since.

During the 1940s the industry exploded with a bang (*not* because of the instability of celluloid) as World War II created a demand for inexpensive, mass-produced products. New materials were invented for the process on a regular basis, and technical advances resulted in more and more successful applications.

CHARTING INDUSTRY EVOLUTION

From its birth in the late 1800s, to recent developments and applications, the injection molding industry has grown at a fast and steady rate. It has evolved from producing combs and buttons to molding products for all production fields, including automotive, medical, aerospace, and consumer goods, as well as toys, plumbing, packaging and construction. Table I-1 lists some of the important dates in the evolution of the injection molding industry.

Table I-1. Evolution of Injection Molding

1868	John Wesley Hyatt injection-molds celluloid billiard balls.
1872	John and Isaiah Hyatt patent the injection molding machine.
1937	Society of the Plastics Industry founded.
1938	Dow invents polystyrene (still one of the most popular materials).
1940	World War II events create huge demand for plastic products.
1941	Society of Plastics Engineers founded.
1942	Detroit Mold Engineering, Inc. (DME) introduces stock mold base components.
1946	James Hendry builds first screw injection molding machine.
1955	General Electric begins marketing polycarbonate.
1959	DuPont introduces acetal homopolymer.
1969	Plastics land on the moon.
1972	The first parts-removal robot is installed on a molding machine.
1979	Plastic production surpasses steel production.
1980	Apple uses acrylonitrile-butadiene-styrene (ABS) in the Apple IIe computer.
1982	The JARVIK-7 plastic heart keeps Barney Clark alive.
1985	Japanese firm introduces all-electric molding machine.
1988	Recycling of plastic comes to age.
1990	Aluminum molds introduced for production injection molding.
1994	Cincinnati-Milacron sells first all-electric machine in U.S.

EVOLUTION OF SCREW CONCEPT AND EVALUATION OF PLUNGER

The machine the Hyatt brothers invented was primitive but performed well for their needs. It was simple in that it acted like a large hypodermic needle and contained a basic plunger to inject the plastic through a heated cylinder into a mold. In 1946, James Hendry began marketing his recently-patented screw injection machine. This auger design replaced the conventional Hyatt plunger device and revolutionized the processing of plastics. Screw machines now account for approximately 95% of all injection machines.

The auger design of the screw creates a mixing action when new material is being readied for injection. The screw is inside the heating cylinder and, when activated, mixes the plastic well, creating a homogenized blend of material. This is especially useful when colors are being molded or when regrind is being mixed with virgin material. After mixing, the screw stops turning and the entire screw pushes forward, acting like a plunger for injecting material into a mold.

Another advantage of screw technology is a reduction of energy requirements. The injection cylinder that holds the plastic being readied for the next cycle features a series of electrical heater bands around the outside. When energized, these bands heat the cylinder to the point of softening the plastic. In addition, because the screw generates friction when it turns within the cylinder, more heat is produced. Thus the material is also heated from the inside out which results in less

heat required from the electrical heater bands to soften the plastic to the correct injection temperature.

Although the screw machine is the most popular, there is still a place for the plunger-type machine. A plunger does not rotate; it simply pushes material ahead, then retracts for the next cycle. It, too, resides within a heated cylinder. Because there is no rotating, there is no shearing or mixing action. So, in a plunger machine the necessary heating action is provided solely by the external heater bands because the plunger produces no friction. Also, if two different colored materials are placed together in the heated cylinder they are not blended together. The plunger simply injects the materials at the same time. If, for instance, the two colors are white and black, the resultant molded part will take on a marbled appearance with definite swirls of black and white throughout the part. This may be a desired finish for particular products, such as lamp bases or furniture, and the use of a plunger machine allows that finish to be molded into the product. Use of a screw machine would result in a single color (gray) product because the two colors would be well mixed prior to injecting.

The injection molding industry has made a huge impact in its short life. Starting in the workshop of the two Hyatt brothers, it has become a major focus for manufacturing of products from toys to medical devices, and most everything in between. The future holds only great promise for more productive, cost-effective methods of producing more products using this technology. Improved methods, materials, processing, and tooling will increase the advantages for product designers and manufacturers who choose plastic injection molding as their primary method of manufacturing.

THE PROCESS

Injection molding is a process in which a plastic material is heated until it becomes soft enough to force into a closed mold, at which point the material cools to solidify and form a specific product. The action that takes place is much like the filling of a jelly donut. A hypodermic-style cylinder and nozzle inject the heated plastic into an opening created in a closed container (mold). The material is allowed to harden again, a finished part is ejected, and the cycle is repeated as often as necessary to produce the total number of pieces required.

Figure 1-1 shows the actual process in simplified form; in actuality, there are more than 100 parameters to be controlled during the process to ensure that a quality part is produced in the most economical way. These parameters are discussed in detail in Volume I of this series, *Plastic Injection Molding...manufacturing process fundamentals*, and should be reviewed by those desiring more information. We highlight some of the parameters in this chapter to better acquaint you with the relationship between the process and the need for proper material selection.

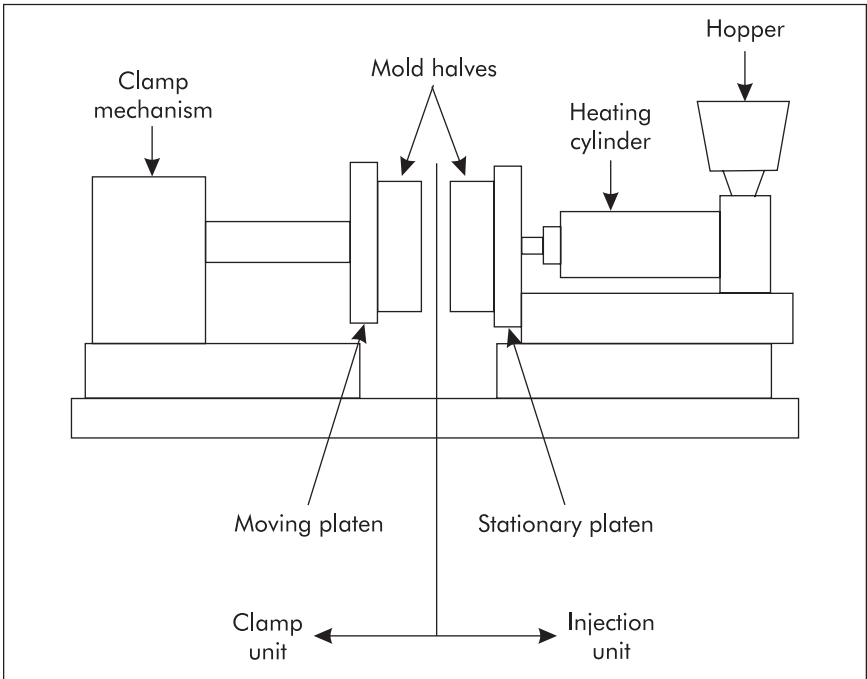


Figure 1-1. The injection molding process.

Categorizing the Parameters

First, we must be aware that, although there are so many parameters to control, they can be detailed within the confines of four major categories. These are *temperature*, *pressure*, *time*, and *distance* as depicted in Figure 1-2.

Note that the circles in the drawing are interconnected and of different sizes. The interconnections indicate that each parameter is both affected by and affects other parameters. A change in one may have a major effect on another. The different circle sizes represent the order of importance placed on each set of parameters; *temperature* and *pressure*, for instance, normally are more important to the process than *time* and *distance*. We take a look at each of the four categories in terms of what's incorporated within each.

Temperature

Temperature of the material. The primary temperature of concern is the temperature to which the plastic material must be heated before it is injected into a mold. All materials have a range of temperatures within which they are most efficiently injected while still maintaining maximum physical properties. For amorphous materials, (those that soften—not melt—when heat is applied) this range

is rather broad; with crystalline materials (those that actually melt when heat is applied) it is fairly narrow. (We discuss the differences between amorphous and crystalline materials in Chapter 2). With both types of materials, however, there is a temperature point at which the plastic flows the easiest and still maintains proper physical properties. This is called the ideal melting point and must be attained through educated guesses and trial-and-error. While this may seem primitive, it is only required as a fine-tuning adjustment once a specific production run is initiated and is finalized as part of establishing particular process specifications for specific products. The guessing process actually begins by setting the temperature of the heating cylinder such that the material being injected is at a temperature recommended for that generic material.

The plastic temperature is measured as it leaves the heating cylinder to make sure it is within the proper range, and then adjusted up or down depending on cycle times, required pressures, mold temperature, and a variety of other parameters. These adjustments are made during a pilot run of the process and until acceptable parts are produced. When parts meet specifications, a setup sheet is created listing the values for all parameters of concern. These values are then stored for use when that specific job is to run again. Table I-2 shows the recommended melt temperatures for some common materials. These are the temperatures that should be referred to when measuring the material as shown in Figure 1-3.

The softening (or melting) of the plastic is achieved by applying heat to the plastic material, causing the individual molecules to go into motion. To a point, the more heat that is applied, the faster the molecules move. However, if too much heat is applied, the plastic material begins to degrade and break down into its main constituents, one of which is carbon.

The heat is applied by electrical heater bands wrapped around the outside of the heating cylinder of the injection molding machine as depicted in Figure 1-4a.

The heater bands, which resemble hinged bracelets, are assembled such that individual groups of three or four control the temperature of a single zone. There are three basic temperature zones for the heating cylinder: *rear*, *center*, and *front*. Each zone is monitored by a thermocouple connected to a temperature controller. The thermocouple determines whether or not the zone is at the correct

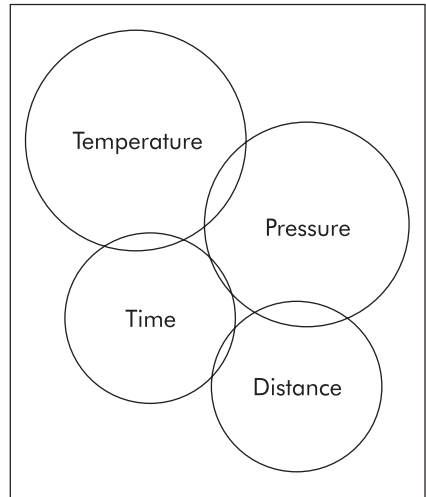


Figure 1-2. Categories of parameters.

Table I-2. Suggested Melt Temperatures for Various Plastics

Material	Temperature, °F (°C)
Acetal (copolymer)	400 (204)
Acetal (homopolymer)	425 (218)
Acrylic	425 (218)
Acrylic (modified)	500 (260)
ABS (medium-impact)	400 (204)
ABS (high-impact and/or flame retardant)	420 (216)
Cellulose acetate	385 (196)
Cellulose acetate butyrate	350 (177)
Cellulose acetate propionate	350 (177)
Ethylene vinyl acetate	350 (177)
Liquid crystal polymer	500 (260)
Nylon (Type 6)	500 (260)
Nylon (Type 6/6)	525 (274)
Polyallomer	485 (252)
Polyamide-imide	650 (343)
Polyarylate	700 (371)
Polybutylene	475 (246)
Polycarbonate	550 (288)
Polyetheretherketone (PEEK)	720 (382)
Polyetherimide	700 (371)
Polyethylene (low-density)	325 (163)
Polyethylene (high-density)	400 (204)
Polymethylpentene	275 (135)
Polyphenylene oxide	385 (196)
Polyphenylene sulfide	575 (302)
Polypropylene	350 (177)
Polystyrene (general purpose)	350 (177)
Polystyrene (medium-impact)	380 (193)
Polystyrene (high-impact)	390 (199)
Polysulfone	700 (371)
PVC (rigid)	350 (177)
PVC (flexible)	325 (163)
Styrene acrylonitrile (SAN)	400 (204)
Styrene butadiene	360 (182)
Tetrafluorethylene	600 (316)
Thermoplastic polyester (PBT)	425 (218)
Thermoplastic polyester (PET)	450 (232)
Urethane elastomer	425 (218)

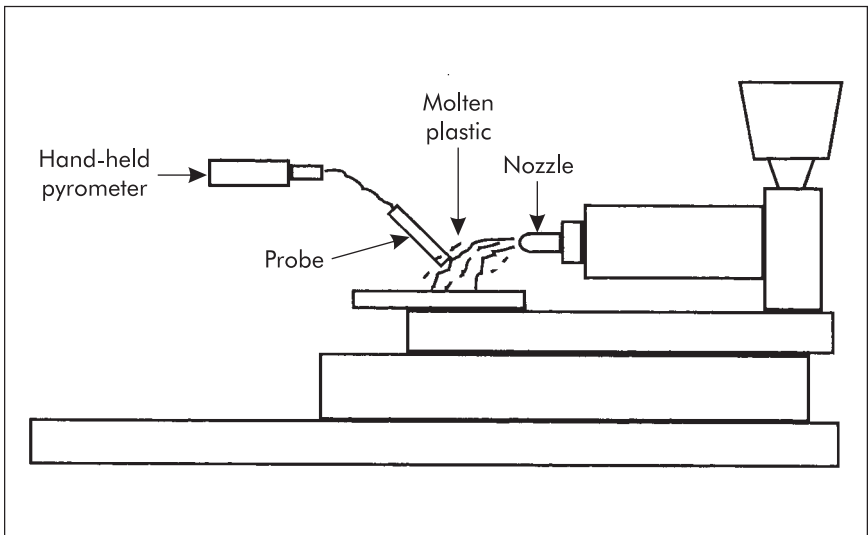


Figure 1-3. Measuring plastic temperature.

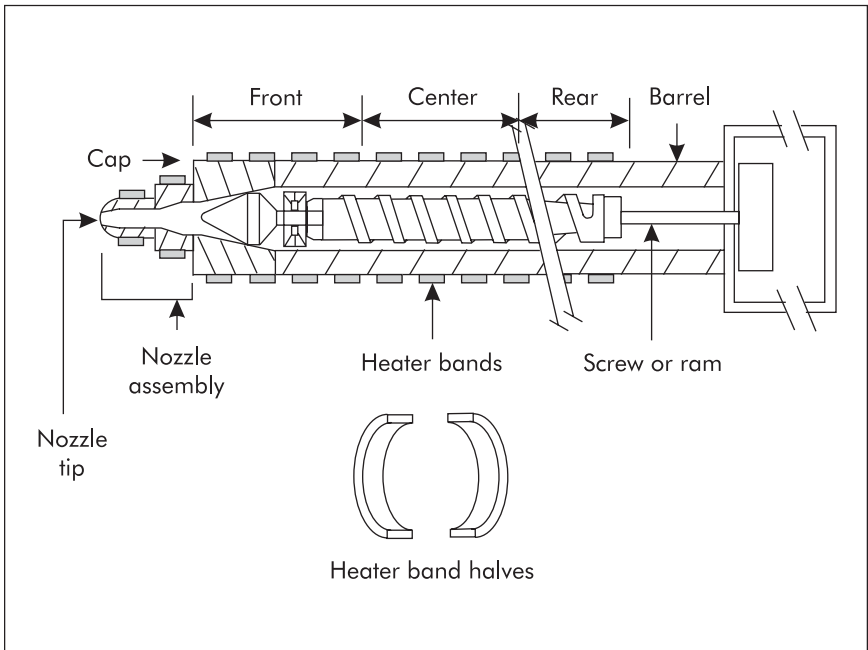


Figure 1-4a. The heating cylinder.

temperature and, if more heat is required, signals the controller to supply more electricity to the heater bands in that temperature zone.

In addition, the machine nozzle (which is mounted at the front of the heating cylinder) incorporates at least one heater and is considered an additional zone called the *nozzle heater zone*. This is depicted in Figure 1-4b.

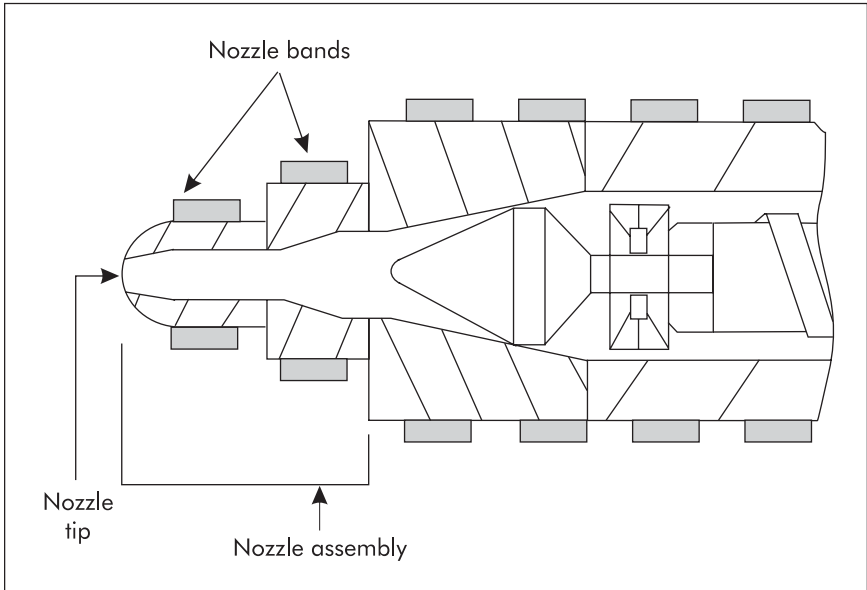


Figure 1-4b. Nozzle heater zone.

Heat is also generated in the heating cylinder by the compressive force of the feed screw turning in the cylinder, Figure 1-5.

This screw augers fresh material into the heating cylinder from the hopper. The turning action squeezes the plastic, thus creating friction, which in turn creates heat. The amount of friction is controlled by a variety of elements such as the rotation speed of the screw, and the distance between the outside diameter of the flights and the body diameter of the screw, which changes along the length of the screw.

Temperature of the mold. Another important temperature is that of the mold. A mold is used for containing the injected plastic in a specific shape while the plastic cools to a solid. After solidifying, the plastic product is ejected from the mold and a new cycle is begun.

The rate at which the plastic cools is an important factor in determining the strength of the plastic material's physical properties, especially with crystalline

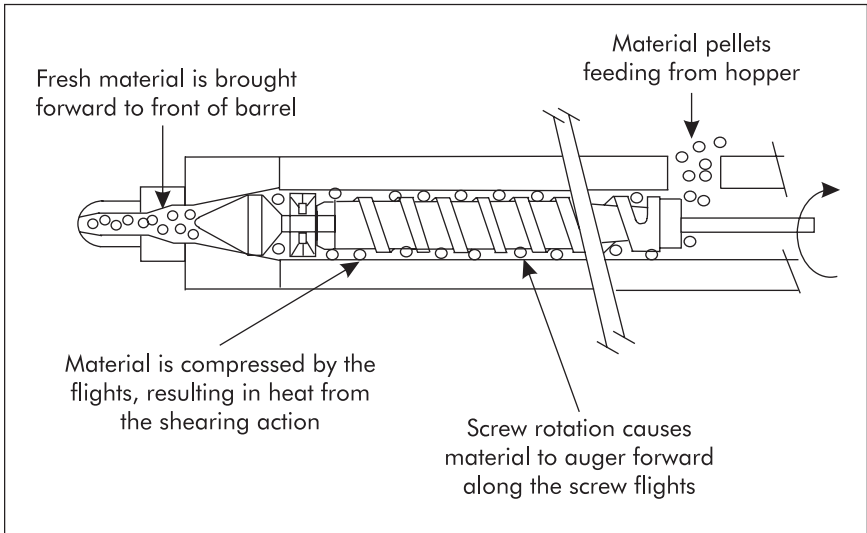


Figure 1-5. Cylinder section showing screw action.

plastics. This is because when the material is first heated, the molecules are disconnected from each other and allowed to move about freely. As the material cools, these molecules must attach themselves to each other again to regain their maximum strength. If they are cooled down too quickly they stop moving before they are fully connected and the result is a product with less than optimum physical strength. So it is important to cool the plastic at a rate slow enough to allow the material to reach proper physical strength, but fast enough to minimize cycle time (and total cost). Table I-3 lists recommended mold temperatures for some common plastics.

Normally, controlling the temperature of the mold is accomplished by running water through specially designed channels machined into the mold. These channels usually consist of a series of holes drilled through specific plates that make up the mold. These holes are connected, by high-temperature hose, to a temperature control unit that supplies the appropriate amount and flow of hot or cold water to maintain a selected temperature (see Figure 1-6).

When hotter temperatures are required, such as at start-up time, the temperature controller cycles the water through a heating device until the proper temperature is reached. When cooler temperatures are required, the unit dumps the circulating water to drain and refills itself with water from the input source. In this manner the unit is capable of maintaining the temperature of the water circulating through the mold to within 2 or 3° F (1.1 or 1.7° C).

Temperature of the oil. Most molding machines are driven by hydraulic systems, although all-electric models are available. These systems provide

Table I-3. Suggested Mold Temperatures for Various Plastics

Material	Temperature, °F (°C)
Acetal (copolymer)	200 (93)
Acetal (homopolymer)	210 (99)
Acrylic	180 (82)
Acrylic (modified)	200 (93)
ABS (medium-impact)	180 (82)
ABS (high-impact and/or flame retardant)	185 (85)
Cellulose acetate	150 (66)
Cellulose acetate butyrate	120 (49)
Cellulose acetate propionate	120 (49)
Liquid crystal polymer	225 (107)
Nylon (Type 6)	200 (93)
Nylon (Type 6/6)	175 (79)
Polyallomer	200 (93)
Polyamide-imide	400 (204)
Polyarylate	275 (135)
Polybutylene	200 (93)
Polycarbonate	220 (104)
Polyetheretherketone (PEEK)	380 (193)
Polyetherimide	225 (107)
Polyethylene (low-density)	80 (27)
Polyethylene (high-density)	110 (43)
Polymethylpentene	100 (38)
Polyphenylene oxide	140 (60)
Polyphenylene sulfide	250 (121)
Polypropylene	120 (49)
Polystyrene (general purpose)	140 (60)
Polystyrene (medium-impact)	160 (71)
Polystyrene (high-impact)	180 (82)
Polysulfone	250 (121)
PVC (rigid)	140 (60)
PVC (flexible)	80 (27)
Styrene acrylonitrile (SAN)	100 (38)
Styrene butadiene	100 (38)
Tetrafluorethylene	180 (82)
Thermoplastic polyester (PBT)	180 (82)
Thermoplastic polyester (PET)	210 (99)

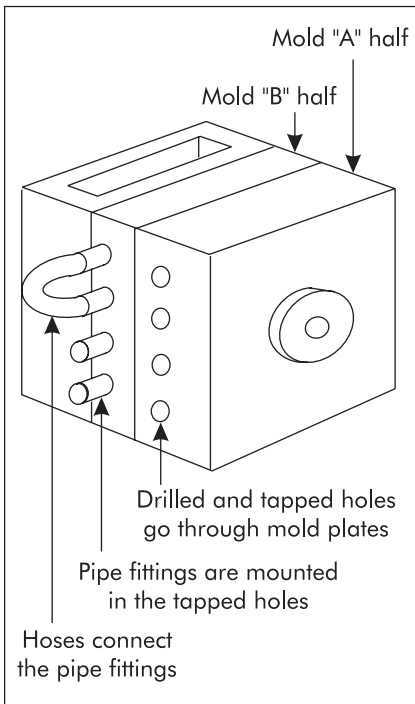


Figure 1-6. Controlling temperature of mold.

energy for, among other things, turning the screw, opening and closing the clamp (even in toggle machines), and actuating ejector systems. During these operations, and even when the machine is running at idle, the temperature of the system's hydraulic fluid rises. This temperature rise results from friction created by the oil flowing through the machine and by the compressive force of the oil when used to provide pressure.

Oil temperature must be controlled to allow it to function properly. If the oil is too cool it will be thick and not flow easily, which may cause valves to operate sluggishly or not at all. If the oil is too hot it will degrade into a thin liquid filled with chunks of additive materials that broke down because of the heat. These clog passageways and interrupt operation of the hydraulic mechanisms.

The temperature of the oil is controlled by a *heat exchanger*. This unit circulates the oil over copper (or other highly-conductive metal) tubing that weaves back and forth inside the unit. The tubing is connected to a water inlet and the temperature of the oil is monitored. When the oil is cold, the heat exchanger does not circulate the water in the tubes. The water absorbs heat from the oil as the oil warms up while flowing through the hydraulic system. When the oil becomes too hot, the heat exchanger opens the water inlet valve to allow water to circulate through the copper tubing. The circulating water draws heat from the oil until the oil temperature drops to the proper level. The system then cycles the water off until needed again. In this way, the heat exchanger can maintain the proper oil temperature at approximately 120° F, $\pm 5^\circ$ F (49° C, $\pm 2.7^\circ$ C).

Pressure

Pressure is required for a variety of reasons in the injection molding process. We will focus on injection pressure, holding pressure, and clamping pressure. Pressure, for these processes, is provided by the hydraulic oil system within the molding machine and a series of control valves, regulators, and directional valves. The system normally provides a primary "line" pressure of 2000 psi (140.6 kg/cm²)

which is then adjusted up or down by the control components of the system to provide whatever pressure is needed for a particular application. For instance, the injection pressure can be adjusted from approximately 500 psi (35.2 kg/cm²) for fast-flowing plastic, up to 20,000 psi (1406.1 kg/cm²) for highly viscous materials. The specific requirements for the various pressure applications follow.

Injection pressure. Injection pressure is the primary pressure used for the injection molding process. It can be defined as the amount of pressure required to produce the *initial filling* of the mold cavity image. (The cavity image is the opening in the mold that will be filled with plastic to form the product being molded.) Note that we use the phrase *initial filling*. Initial filling represents approximately 95% of the total filling of the cavity image.

The amount of pressure required will range from very low (500 psi [35.2 kg/cm²]) to very high (20,000 psi [1,406.1 kg/cm²]). What determines how much pressure is used is the viscosity and flow rate of the plastic being injected. Of the more than 20,000 plastic materials available today, most will fall within a pressure range requirement of approximately 5000 to 15,000 psi (351.5 to 1054.6 kg/cm²). And most of these will have a flow index rating of between 5 and 20.

The flow index rating (properly referred to as the melt index) is a value designating the amount of material in grams that flows over a 10-minute period from a specially designed rheometer. This is an official ASTM standard, number D-1238, and is depicted in Figure 1-7.

The melt index (MI) test is a good reference test to determine the relative thickness (viscosity) of a plastic material, and thereby its relative ease of flow during the injection process. An MI of around 5 indicates a very viscous material and requires molding pressures that are very high. On the other hand, materials with an MI of 20 are easy to mold and require low pressures. The materials data sheet supplied by the manufacturer indicates the melt index range for a specific material.

Of importance here is that the melt index also indicates the properties of the plastic material being tested. Table I-4 shows some of these indications.

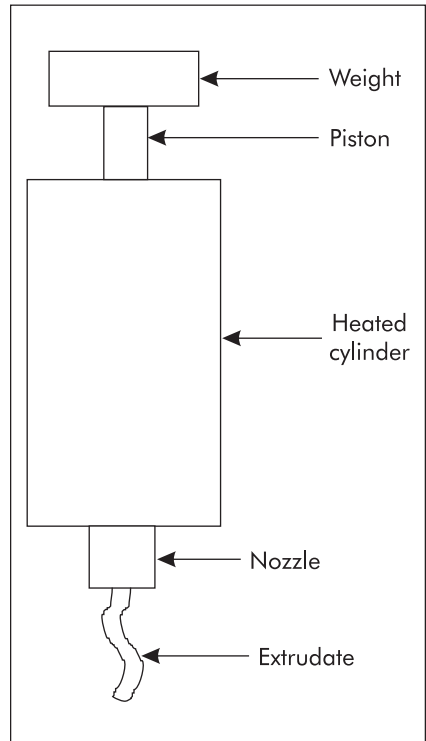


Figure 1-7. Melt index rheometer.

Table I-4. Melt Index Value's Impact on Plastics Properties

As melt index value *decreases*

Stiffness	Increases
Tensile strength	Increases
Yield strength	Increases
Hardness	Increases
Creep resistance	Increases
Toughness	Increases
Softening temperature	Increases
Stress-crack resistance	Increases
Chemical resistance	Increases
Molecular weight	Increases
Permeability	Decreases
Gloss	Decreases

Note that permeability and gloss actually *decrease* as the melt index value drops.

This chart shows the relative importance of increasing or decreasing the melt index value. A customer can request an increase or decrease from the material supplier as long as the requested value is within the range of values for that specific material. In most cases, a stiffer material is preferred, but it will be harder to process because it will require higher injection pressures. A major drawback though, other than cost, is the introduction of stress caused by using the higher pressures. This is discussed in a later chapter, but be aware that stress is molded into every part. The degree of stress can be controlled, however, and should be minimized whenever possible to eliminate defects and field failures of a specific product.

Holding pressure. Holding pressure is applied at the very end of the primary injection stroke and is used for the final 5% filling of the cavity image. It is called holding pressure because it *holds* pressure against the cooling plastic in the cavity image while that plastic solidifies. This helps to ensure a dense part, molded with uniform pressure and controlled shrinkage. Holding pressures are usually in the range of 50% of the primary injection pressure. So, if a plastic material requires 10,000 psi (703.1 kg/cm²) for primary injection pressure, the holding pressure should be applied at approximately 5,000 psi (351.5 kg/cm²). Holding pressure must be applied against a pad of extra material called a *cushion*. This is discussed in more detail later in this chapter.

Clamp pressure. Clamp pressure can be defined as the amount of pressure required to hold the mold closed against injection pressure. The clamp unit of a molding machine can be mechanically or hydraulically activated, and this

pressure is applied against the mold that forms the plastic product. The degree of pressure applied must be at least equal to the amount of pressure applied by the injection unit. If 10,000 psi (703.1 kg/cm²) *injection pressure* is used, then at least 10,000 psi *clamp pressure* must be used. In fact, a precautionary measure of additional pressure equal to approximately 10% should be used to ensure that the clamp stays closed in the event that the injection pressure drifts upward slightly.

If the clamp pressure is too low, the mold will blow open during injection. Flash will occur and the cavity image will not fill with plastic. If the clamp pressure is too great, the mold may collapse from the total force being applied.

Time

Injection time. The amount of time required for injection activities depends on how much material is being injected, the viscosity of the material, and the percentage of the machine's barrel capacity that is being utilized. The total amount of injection time required is divided into two separate phases: *initial* injection and *holding* injection.

Initial injection time. When the mold closes completely, either a limit switch or pressure buildup (or both) signals the injection screw to push forward and inject the molten plastic into the closed mold. The screw does not turn at this point but acts only as a plunger to force the material into the mold. This initial injection is performed using the highest practical pressure for the specific application (normally 10,000 to 15,000 psi [703.1 to 1054.6 kg/cm²]) in the fastest practical amount of time. In most cases this is less than 2 seconds and rarely more than 3 seconds. Sometimes, depending on machine design, this initial action is divided into two or three smaller actions, or *stages*. In those cases, the total amount of injection time normally does not exceed 4 or 5 seconds. The initial injection time is controlled by way of a timer. If a "booster" injection time is available, it will be included in the first stage of the initial injection time. When a booster phase is used, the injection machine's entire hydraulic system (injection and clamp) is combined to push a large volume of oil through the system. This can greatly increase the speed at which the material is injected.

Injection hold time. On most machines, the timer for initial injection time (also called injection forward time) controls the total amount of time the injection screw is pushing forward. The initial injection time is the first part of that time, and injection hold is the latter part of that time. On some machines the hold time and initial time are on separate timers.

The hold time is the amount of time the injection screw maintains pressure against the plastic after it has been injected into the mold. This pressure is applied against the "cushion" (or pad) and is applied long enough for the gate to "freeze off." Freezing off means to solidify. Visualizing the plastic being

injected into the mold, it can be determined that the molten plastic enters the mold cavity image through a gate. This gate is the first point at which the plastic actually “sees” the cavity image. When all the required material goes through the gate and packs the cavity image, the plastic is allowed to cool, under hold pressure, down to the point at which it all solidifies. But because it is normally the thinnest part of the cavity image, the gate is the first thing to solidify. Once it does, there is no reason to keep pressure because the plastic in the cavity lies beyond the solidified gate and the pressure from the injection unit no longer has any effect on it. So the length of time required to hold pressure against the gate is only long enough for the gate to freeze (solidify). In most cases this is only a matter of a few seconds. A gate with a thickness of 0.062 in. (1.57 mm) would take approximately 6 seconds to solidify.

Cooling time. Cooling time is probably the most important time phase in the entire injection process. Cooling time is the amount of time required for the plastic material to cool to the point at which it has (1) solidified, and (2) become rigid enough to withstand the ejection process. Ejection of the part is the process that pushes the finished molded product out of the mold after the entire cycle is completed. Even though the plastic may cool enough to solidify, it may not be rigid enough to eject. This is because the cooling process actually takes as long as 30 days to finalize. The initial cooling is rapid, and 95% of the total cooling takes place in the mold. But the other 5% takes place outside the mold. If the outer skin of the plastic product is solidified to a sufficient depth, the remaining cooling will not have an appreciable effect on the molded part. But if the skin is too thin, the remaining cooling will cause shrinkage stress to build up and the molded part may warp, twist, blister, or crack.

The key to minimizing these problems is to keep the part in the mold for a sufficiently long period of time, but no longer than necessary because time is money, and long cycles are expensive. Most material suppliers are more than happy to share cooling time requirements for their specific materials at varying thicknesses (the thicker the part the longer the cooling time required), but, on the average, a 0.062 in. (1.57 mm) -thick wall should take approximately 9 to 12 seconds to solidify (depending on material) to the point at which it can be ejected from the mold without undue distortion.

Clamp time. Clamp time (or “mold closed” time) is the amount of time that the mold halves stay closed, or clamped together. It begins when the mold first closes and continues until the mold opens again. All of the injection and material preparation phases take place during the clamp time, as does the cooling time. So the clamp time is the greatest amount of time involved with injection molding, since it encompasses most of the operations that take place during the injection molding process.

Distance

The final parameter is that of distance. Although it's the last item on the list of parameter priorities, control of distances is critical to producing high-quality products at reasonable cost. This is primarily due to the fact that excessive distance requires excessive time, and, as stated earlier, time is money. Because distance is so closely related to time, the various functions involving distance are basically the same as those related to time, plus a few others.

Mold-close distance. There are two phases to the mold-close distance: the "initial close," covering the major portion of the closing process, and the "final close," covering the very last portion of the closing process.

The distance covered by the initial mold closing should take the mold halves to within approximately 1/4 to 1/2 in. (6.4 to 12.7 mm) from touching. This closing distance is normally traveled at high speed and is done as an effort to get the mold closed as soon as possible so the overall cycle time can be minimized. But if the mold halves are simply allowed to slam together under that high speed, they will eventually crack and break. Also, there may be an obstruction in the mold (such as a broken piece of plastic) which will cause damage if the mold is allowed to close upon it. So although the mold closes quickly, it closes under very low pressure, and then only until the mold halves are close to touching. The closing distance is measured from the point at which the mold begins to close, to the 1/4- to 1/2-in. gap mentioned above (see Figure 1-8).

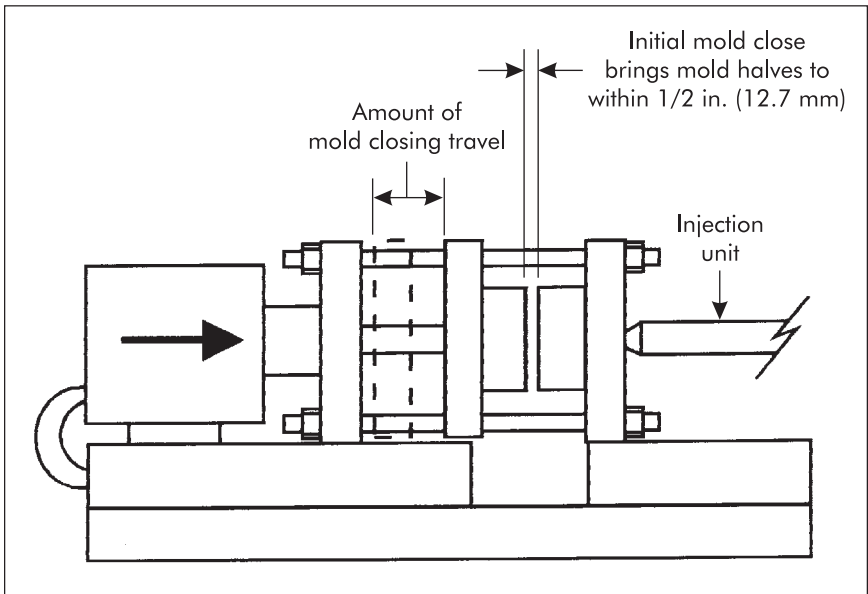


Figure 1-8. Mold-closing distances.

At that point, the mold is slowed to a crawl, but placed under high closing pressure. This occurs for the entire 1/4- to 1/2-in. distance of travel, until the mold halves are closed tightly against each other. This slow final closing is done to minimize closing damage. If a foreign object is caught in the mold, the closing action will stop at this point and the object can be removed without causing damage. If the clamp is closed under high pressure any foreign object will be crushed, resulting in damage to the mold.

After traveling the final 1/4 to 1/2 in., the mold is fully closed under full clamp tonnage and the injection phase is allowed to begin.

Injection distance. As mentioned, the injection process is performed in at least two phases. The first is *initial* injection and the second is *injection hold*.

Initial injection distance. Initial injection distance (Figure 1-9) must be set to ensure that approximately 95% of the intended material is injected. This distance varies depending on how big the machine is and how much of the barrel capacity is being injected for one shot. With the ideal shot size being 50% of the barrel capacity, a machine rated as having a 6-oz (170-g) barrel capacity (in styrene), would have an ideal single shot size of 3 oz (85 g). The limit switch governing this shot size would be set half-way back on the measurement scale. This scale is usually located on the injection barrel but sometimes is part of the electronic control system. In any case, the scale is incremental and can be adjusted anywhere between 0 and 100% of barrel capacity.

Holding distance. With the initial injection setting allowing 95% (approximately) of the material to be injected, the machine switches to holding pressure.

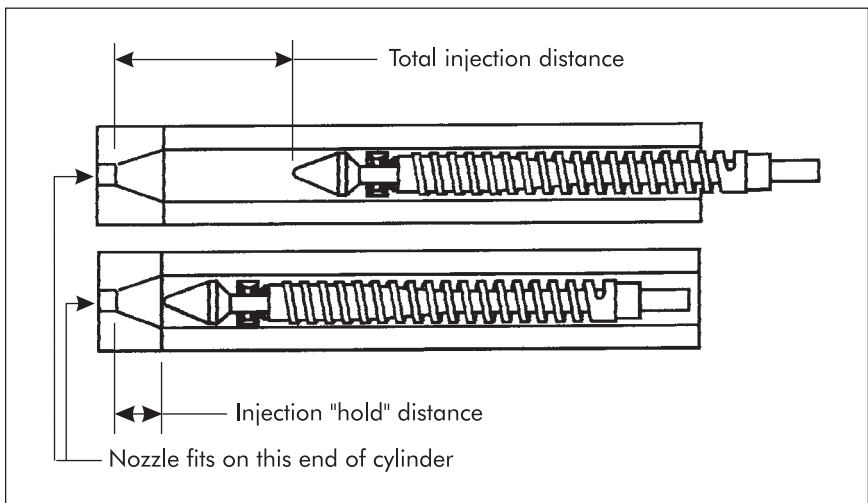


Figure 1-9. Injection and hold distances.

This finishes filling the mold and holds pressure against the injected material. The point at which the hold pressure takes over should be set almost at the very end of the stroke of the injection screw.

Cushion (pad). A pad or cushion of material (between 1/8 and 1/4 in. [3 and 6 mm] thick) should be left in the barrel for hold pressure to be applied against, as shown in Figure 1-10.

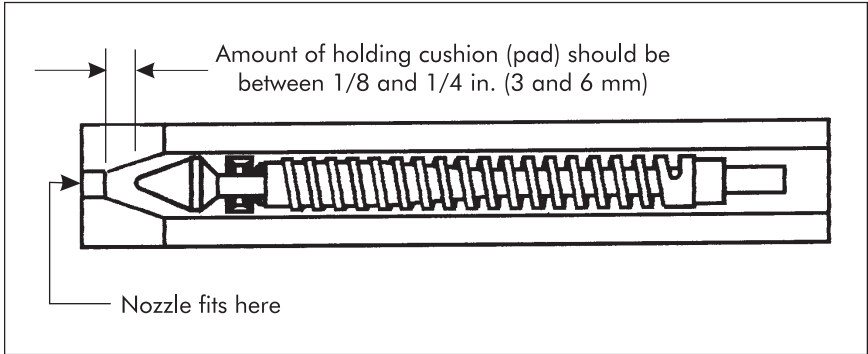


Figure 1-10. Cushion.

The amount of cushion is established by creating a total shot size slightly larger than that required to fill the mold. For example, if the amount of material required to fill the mold is 2.9 oz (82 g), the total shot size would be established at approximately 3.0 oz (85 g). This would then be increased or decreased during setup until the 1/8-in. (3-mm) cushion is developed. This adjustment is made by changing the set point for the screw return.

The thickness of the cushion is critical. It must not be less than 1/8 in. (3 mm) because anything less is difficult to control accurately and there is a good chance that the cushion will go to zero on a random basis due to inconsistencies in the specific gravity of the melt. If the cushion does go to zero (“bottoms out”), there will be no pressure against the material in the mold and the molded part may warp, crack, or simply result in a nonfill due to the absence of holding pressure. Moreover, the shrinkage will vary and the part may not be dimensionally acceptable.

If the cushion is more than 1/4 in. (6 mm) thick, the plastic in the cushion might cool down and begin to solidify due to all the steel surrounding it, blocking the nozzle, and resulting in slow flowing material or no flow at all.

Mold-open distance. To break the vacuum created in the cavity image during the injection process, the mold must be opened slowly. After the mold has opened 1/4 in. (6 mm) or so, the vacuum on the stationary side is relieved and the mold

can be allowed to open the rest of the way at a faster rate. The faster rate is desired so that the cycle can be completed quickly and the next cycle begun.

If a mold contains “actions,” such as slides or cams, there may be a requirement to open the mold slowly for a longer distance. This might range from the original 1/4 in. (6 mm) to 2 or 3 in. (51 or 76 mm) or more.

Once the mold has opened enough to break the vacuum (and far enough to clear the actions) it can be opened fully. The total distance a mold is to open should not be more than what is absolutely necessary because it takes time to open the mold, and time is money. So, how much is necessary?

The mold should open a total distance equal to twice the depth of the molded part. For example, if the part being molded is a square box with a depth of 6 in. (152 mm), the mold should be allowed to open no more than 12 in. (305 mm). If possible, this dimension should be made smaller. All that is needed is enough open space to allow the finished part to fall clear of the mold after ejection or for the operator to reach in and remove the ejected part (see Figure 1-11).

If the mold is running with an operator who must manually remove the part from the ejector system, the mold-open distance should be adjusted to whatever level is necessary from the standpoint of safety and comfort, in addition

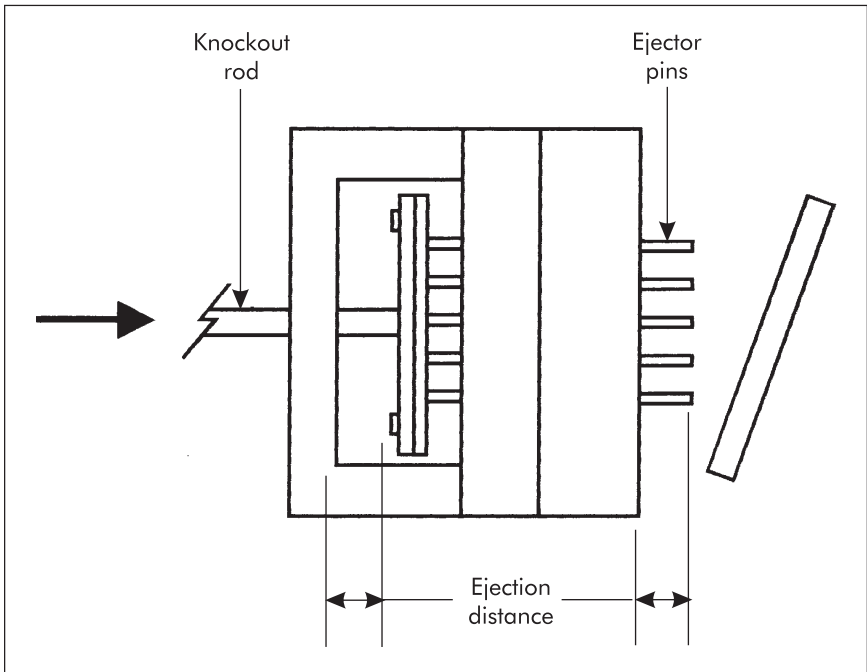


Figure 1-11. Ejection of finished part.

to simply allowing enough room for manipulating the part. In most cases this will not exceed 2 1/2 times the depth of the part.

Ejection distance. The only area of the part that is ejected is that which is confined in the moving half ("B" half) of the mold. If those areas have a maximum depth of 1 in. (25 mm), then theoretically only 1 in. of ejection is required. If more ejection distance is used, it takes additional time, and time is money. If less is used, the part will probably not fall free and will get stuck, causing damage if the mold closes on it.

It is a good practice to measure how much ejection is required and then add 1/8 to 1/4 in. (3 to 6 mm) to make sure the part is well clear of the mold surface.

HORIZONTAL VERSUS VERTICAL MOLDING

Although there are advantages to both horizontal and vertical molding, the standard method of injection molding utilizes a machine that opens and closes the mold along a horizontal path, Figure 1-12. The injection unit is on that same horizontal path.

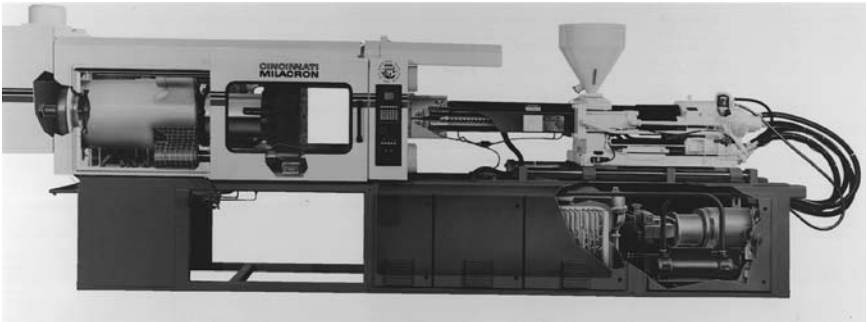


Figure 1-12. Horizontal molding machine. (Courtesy Cincinnati Milacron)

A vertical molding machine, Figure 1-13, opens and closes the mold along a vertical path. The injection unit stays on the horizontal path.

Advantages of Horizontal Molding

- Horizontal molding is the most common method of injection molding. Therefore, large numbers of experienced personnel are available.
- Training videos, handbooks, and troubleshooting media are readily available for horizontal molding.
- Gravity assists ejected parts to fall freely from the mold.



Figure 1-13. Vertical molding machine. (Courtesy Van Dorn Demag Corp.)

- The mold's sprue bushing can stay in contact with the injection unit, thus maintaining heat stabilization.

Advantages of Vertical Molding

- Vertical molding machinery requires less floor space because the clamp unit is vertical.
- Vertical molding is normally used for shuttle molds where one set of parts is being molded while another is being prepared with inserts or overmolded parts, thus minimizing gate-to-gate cycle times.
- Vertical molding allows easy operator access to molded parts. This can be important when molding fragile products that should not be allowed to fall freely as in horizontal molding.

- Vertical molds are normally less complicated than horizontal molds and can be built at less expense.

THE NEED FOR PROCESS CONTROLS

To take advantage of the numerous benefits of injection molding, it is necessary to control as many parameters of the process as possible. Actually controlling those parameters can be fairly simple. But is it really necessary? Should valuable resources (time, money, personnel, space) be allocated to this end? The answer to both questions should be a resounding *Yes!* The reason is that any successful method of controlling the quality and the cost of a product depends heavily on consistency of the process used to manufacture that product. Consistency can only be achieved by tightly controlling as many manufacturing process parameters as possible. This does not mean that adjustments cannot be made as the job is running. It simply means that proper control allows for accurate, meaningful adjustments when they are necessary. If proper control of parameters is attained, consistency follows. This consistency takes the form of part quality and part cost.

Part Quality

Part quality requirements are usually determined through joint discussions and agreements between the owner of the product design and the manufacturer of the product. This may be the same person (or group), or two individual entities, and in some cases the owner of the product design sets down the requirements without any discussion with anyone else. In any case, a set of requirements is established that determines how the finished product will respond to a given set of circumstances. These requirements include mechanical, physical, aesthetic, thermal, and/or other values, and tolerances are placed on each of these values, if practical. For example, the product designer may wish the finished product to withstand high temperature exposure for extended periods of time. Or there may be a requirement that the product be a specific length. Both of these requirements must have a value placed on them. The temperature requirement might be listed as 160° F (71° C), and the length requirement might be 10 in. (254 mm). Reasonable tolerances are placed on these values and the manufacturer (molder) then understands the relative importance of those two requirements. For instance, the 160° F becomes 160° ± 10° F (± 5.6° C), and the 10 in. becomes 10 ± 1/2 in. (12.7 mm).

Understanding the importance of the requirements dictates to the molder how much control is essential to achieve the necessary level of consistency to meet these requirements at the most efficient and economical level of manufacturing. The molder knows, for instance, that the 10-in. (254-mm) length dimension

can be made to be slightly larger if high injection pressures are used, or slightly smaller if low injection pressures are used. But the molder must be aware that the closer that dimension must be held, the tighter the control must be maintained on the pressure parameter. This does not say that all parts must be molded to extremely high quality levels. It simply states that once the level of quality required is understood, it can be met and maintained through proper control of molding parameters. *The lower the level of quality required, the easier it is to control the processing parameters.*

Part Cost

The cost to mold a part is determined in a variety of ways and the methods are discussed in Volume I of this series. Once that cost is estimated (and successfully quoted) it becomes the responsibility of the molder to ensure that the cost is not exceeded. Actually, the molder should try to reduce the true cost if possible, and this reduction can be passed on to the customer, kept as additional profit by the molder, or shared equitably by both. Proper control of the process parameters will allow cost maintenance (or reduction) to occur.

It is important to remember that every part molded, whether good or bad, is bought by someone. The customer buys the good parts, and the molder buys the bad parts. So it makes good sense for the molder to strive for zero-defect production, and to keep manufacturing costs to a minimum. This holds true whether the molder is making inexpensive flower pots or precision-molded electronic devices. There is no good reason for the molder to produce bad parts that the customer will not buy.

Producing consistently acceptable products, at the lowest possible cost, can be done by maintaining consistent processing cycles through consistent parameter control. Let's look at some of the concepts involved in accomplishing that.

PARAMETER EFFECTS

Table I-5 shows some of the property values that can be adjusted by a plus or minus change in some of the more common molding parameters.

These are just some examples. But notice how some properties are changed in the same way by different parameters. For instance, "less shrinkage" can be attained by either increasing injection pressure or increasing mold temperature, and "less degradation" can be achieved by lowering back pressure as well as lowering melt temperature. These examples demonstrate that the basic molding parameters do work closely together, and that changing a parameter in one area may affect a value of some property in another area. By understanding this relationship, it is possible to minimize the number of adjustments required when it is necessary to make a correction due to an unexpected change in some variable of the process.

Table I-5. Parameter Change versus Property Effect

Parameter	Property Effect
Injection pressure (+)	Less shrinkage, higher gloss, less warp, harder to eject
Injection pressure (−)	More shrinkage, less gloss, more warp, easier to eject
Back pressure (+)	Higher density, more degradation, fewer voids
Back pressure (−)	Lower density, less degradation, more voids
Melt temperature (+)	Faster flow, more degradation, more brittle, flashing
Melt temperature (−)	Slower flow, less degradation, less brittle, less flashing
Mold temperature (+)	Longer cycle, higher gloss, less warp, less shrinkage
Mold temperature (−)	Faster cycle, lower gloss, greater warp, higher shrinkage

So what are the proper parameter values? In other words, what's the best setting for the injection pressure, back pressure, melt temperature, mold temperature, etc.? It all depends on the material being molded and the type of mold being used, as well as the status of the injection machine and environmental conditions.

THE SETUP SHEET

It is necessary to understand that, in most molding facilities, it is common to have a setup sheet that lists a variety of the common parameters and the value for each. This is usually used to start up a mold at the beginning of production. A typical setup sheet format is shown in Figure 1-14.

Notice in the areas identified as "Temperature," "Timer Settings," "Pressure Settings," and "Miscellaneous," two columns are provided for listing values. The first is marked Start-up, the second Actual, which makes this type of form quite helpful because it acknowledges that there should be one group of settings for start-up, and another for actual long-term production.

Both sets of values are needed because the parameters set for the initial parts will begin affecting each other immediately after starting up. They will eventually stabilize (after approximately 6-8 hours), and at that point they will be affecting the molding material in a different way from the initial start-up effects. For instance, if the "barrel rear temperature" is set to 475° F (246° C) at the beginning of a run, it takes approximately 45 minutes for the material inside that barrel zone to get close to that setting. Then the temperature controller begins to cycle off and on to maintain the temperature properly. This results in a "soaking" of the barrel until the entire zone is at the proper temperature. In the meantime, the material going through that zone has been heated in different stages, with the first mass of material being heated to approximately 450° F (232° C), the next few batches being heated to approximately 460° F (238° C), and the final batches being heated to approximately 470° F (243° C), before the next, and continuing material batches

ABC MOLDING COMPANY			SETUP SHEET			JOB# _____		
Customer Name _____			Part Number _____			Date _____		
Machine Data				Timer Settings				
Preferred Machine _____				<input type="checkbox"/> Automatic <input type="checkbox"/> Semiautomatic <input type="checkbox"/> Manual				
Clamp Tonnage Required _____				Area Start-up Actual				
Clamp Stroke _____				Mold Open _____				
Misc. Equipment Required _____				Injection Delay _____				
_____				Injection Fwd _____				
_____				Injection 1st _____				
_____				Injection 2nd _____				
_____				Injection Hold _____				
				Mold Close _____				
				Decompress _____				
				Ejectors Fwd _____				
				Air Blow Off _____				
Mold Data				Pressure Settings				
Mold Number _____				Area Start-up Actual				
Part Description _____								
No. of Cavities _____								
Shut Height _____								
Mold Open Distance _____								
Runner Type _____ Clamp _____								
Nozzle Type _____				Accumulator _____				
Ejection Stroke _____				Back Pressure _____				
Special Requirements _____				Injection 1st _____				
_____				Injection 2nd _____				
_____				Injection Hold _____				
_____				Ejection _____				
Temperature				Miscellaneous				
Area Start-up Actual				Area Start-up Actual				
Feed Throat _____				Overall Cycle _____				
Barrel Rear _____				Cushion (in.) _____				
Barrel Center _____				Screw rpm _____				
Barrel Front _____				Clamp Speed _____				
Nozzle _____				Ejector Speed _____				
Mold "A" Half _____				_____				
Mold "B" Half _____				_____				
Hot Runner Zone Settings				_____				
1 _____	2 _____	3 _____	4 _____	_____				
5 _____	6 _____	7 _____	8 _____	_____				
9 _____	10 _____	11 _____	12 _____	_____				
Material Description _____				Prepared by _____				
Material Number _____				Date _____				
Total Shot Weight _____				Approved by _____				
				Date _____				

Figure 1-14. Typical setup sheet.

are heated to the correct temperature. (This holds true even if the material is allowed to sit and soak for 30 minutes before beginning the run. Once the run starts, all the parameters begin the stabilization process.)

While all of this is going on, the operator will have adjusted the major parameters to accommodate the original batches of material, so a parameter such as injection pressure would be set high enough to inject that material into the mold. After a few minutes, when the next few batches come through (at a higher temperature) they flow easier and tend to begin flashing if the injection pressure is still at the higher level. And, finally, when the properly heated batches come through, they are flowing so easily that they will, in fact, flash. Then, injection pressures must be adjusted downward to accommodate the hotter material. So by the time everything stabilizes, which may take 6 hours or more, most of the parameters have to be reset to different values from those at which they started. This explains the reason for maintaining two separate parameter sheets or, at least, two separate columns of data on a single parameter sheet.

CONTROLLING SHRINKAGE

All materials have a specific shrinkage rate value assigned by the material manufacturer. The use of the term “rate” is actually a misnomer because it implies that the shrinkage occurs as a function of time, which is not true. Despite this inaccuracy, we use the term because it has become accepted throughout the industry. Shrinkage rate is a value used to predict how much difference there will be between the plastic product when it is first molded and when it has cooled to room temperature.

Everything except water expands when it is heated and shrinks when it is cooled and plastic material is no exception, as shown in Figure 1-15. Each plastic material has a distinct value for how much it will shrink after it is heated and then allowed to cool. This value is referred to as the shrinkage rate and is listed as so many “inches per inch” (in./in.) (mm/mm). It means that for each inch of dimension on the plastic product, the material will shrink a certain number of inches. Usually, plastic materials shrink somewhere between 0.000 in./in. up to approximately 0.050 in./in. (1.3 mm/mm). Shrinkage also can be thought of in terms of percentage. A part that shrinks 0.010 in./in. (0.25 mm/mm) will shrink a total of 1%. One with a shrinkage rate of 0.020 in./in. (0.51 mm/mm) will shrink 2%. One with a 0.005 in./in. (0.13 mm/mm) rate will shrink 1/2 of 1%.

Taking a shrinkage of 0.010 in./in. (0.25 mm/mm) we will see the effect on a part that is 6 in. (152 mm) long. Remembering that the shrinkage is for each inch of product, we would multiply the shrinkage rate times 6 in. That gives us a total of 0.060 in. (1.5 mm) of shrinkage for that single dimension (0.010×6). When a moldmaker builds the cavities that will form the finished plastic product, he or

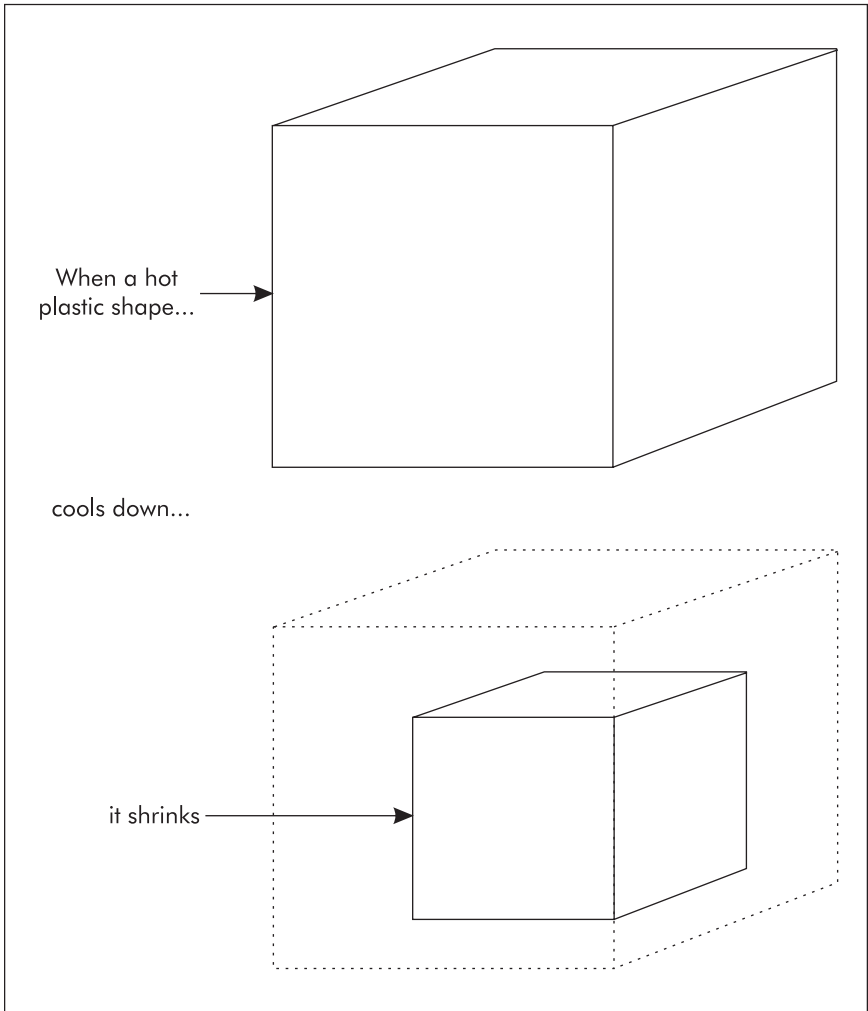


Figure 1-15. Illustrating shrinkage rate.

she must allow for that shrinkage. So the moldmaker would make the steel that will form the 6-in. dimension, 6.060 in. (153.9 mm). Then, when the material cools, it will shrink down to the desired 6-in. dimension.

Note in Figure 1-16 that the mold cavity dimension for the length of the plastic ruler is 6.060 in. (154 mm) and the width is 0.505 in. (12.8 mm). Assuming that shrinkage is the same in all directions, the plastic that fills that cavity will shrink to 6 in. (152.4 mm) by 0.500 in. (12.7 mm) when it cools, because it has a shrink rate of 0.010 in./in. (0.254 mm/mm).

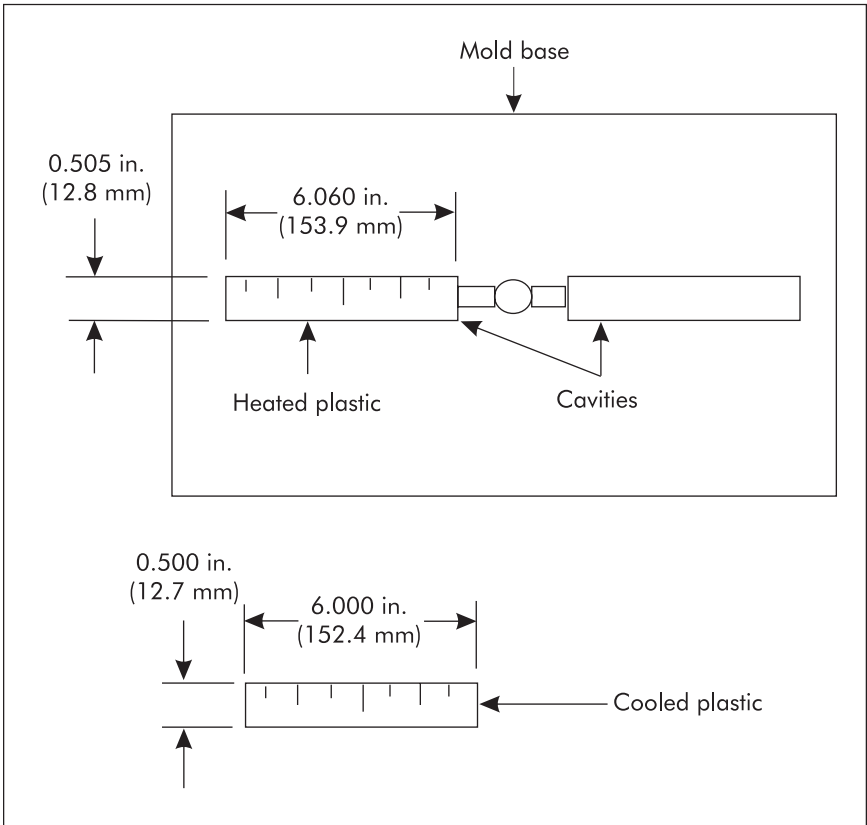


Figure 1-16. How shrinkage affects dimensions.

All plastics are generally categorized as having either low, medium, or high shrinkage. Low shrinkage is within a range of 0.000 to 0.005 in./in. (0.000 to 0.127 mm/mm). Medium shrinkage is within a range of 0.006 to 0.010 in./in. (0.152 to 0.254 mm/mm). High shrinkage is anything over 0.010.

It is important to understand that shrinkage differs between amorphous and crystalline materials, as shown in Figure 1-17. Amorphous materials tend to have low shrinkage rates and the shrinkage occurs equally in all directions. This is called *isotropic shrinkage*. Crystalline materials tend to have high shrinkage rates and the shrinkage is greater in the direction of flow than across the direction of flow. This is called *anisotropic shrinkage*. When using reinforced materials, however, shrinkage will be less in the direction of flow and more across the direction of flow. This is due to the orientation of the reinforcement fibers.

Because of the inherent differences between amorphous material shrinkage and crystalline material shrinkage, there is a greater range of shrinkage control

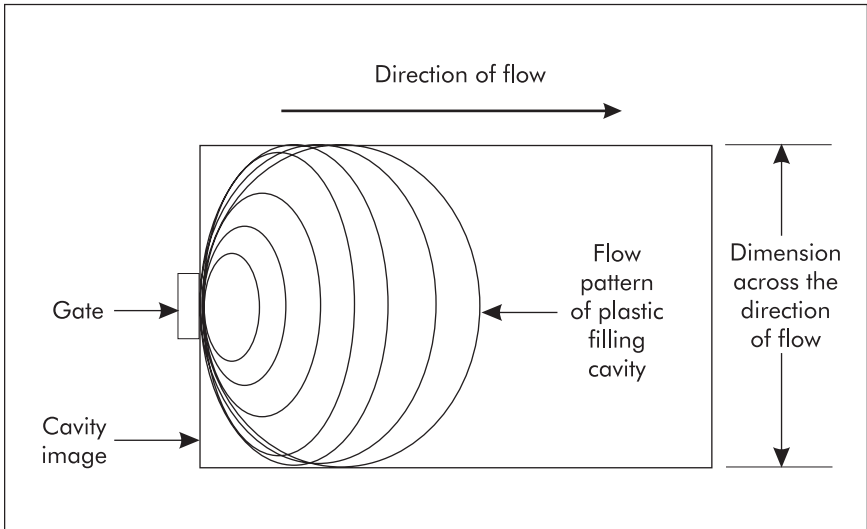


Figure 1-17. Amorphous versus crystalline shrinkage differences.

for amorphous materials. Crystalline materials tend to have higher shrinkage rates in general, but have much less response to processing parameter changes toward shrinkage control. The following information, while general, applies more towards amorphous materials than crystalline.

The Effects of Temperature Adjustments

One way of altering the amount of shrinkage for a specific product or material is to adjust the temperature of the plastic while it resides in the barrel. In general, the higher the plastic temperature, the greater the amount of shrinkage. This is because of the activity of the individual plastic molecules; as the temperature rises, these molecules expand more and take up more space. The higher the temperature, the greater the expansion. The reverse of this is also true; the lower the temperature, the lower the degree of expansion, therefore the lower the amount of shrinkage as the plastic cools.

A general rule of thumb is that shrinkage rates can change 10% by changing barrel temperatures 10%. Thus, if a material exhibits a shrinkage rate of 0.005 in./in. (0.127 mm/mm) at a barrel temperature of 500° F (260° C), it can be lowered to 0.0045 in./in. (0.114 mm/mm) or raised to 0.0055 in./in. (0.140 mm/mm) by altering the barrel temperatures to 450 or 550° F (232 or 288° C), respectively. These are extreme changes and may not be practical for other reasons, but they do represent the 10% rule of thumb.

Shrinkage can be adjusted by altering temperatures of the mold also. A hot mold will create less shrinkage than a cold mold, because the cold mold solidifies

the plastic “skin” sooner than a hot mold, resulting in a shrinking of plastic before full injection pressure is applied. On the other hand, a hot mold allows the molecules to continue to move and be compressed by injection pressure before solidifying. This results in less shrinkage because the molecules are not allowed to move as much after solidifying. A rule of thumb here is that a 10% change in mold temperature can result in a 5% change in original shrinkage.

The Effects of Pressure Adjustments

Injection pressure has a direct effect on shrinkage rates. The higher the injection pressure, the lower the shrinkage rate. This is because the injection pressure packs the plastic molecules together. The higher the pressure, the tighter the molecules are packed. The more they are packed, the less movement they are allowed as they are cooled. This lower movement results in lower shrinkage. The pressure rule of thumb is that a 10% change in pressure can cause a 10% change in shrinkage rate. Of course, the shrinkage is controlled only for as long as the pressure is applied. As long as the pressure is maintained to the point that the plastic cools to solidification, the shrinkage will be controlled. If the pressure is relaxed before that point, the shrinkage will increase because the molecules have been allowed to move again.

Post-mold Shrinkage

There is a delicate balance between maintaining the quality of a molded product and reducing the cost of molding that product. Controlling the shrinkage is only a part of that battle, but it should be understood that the lower the desired amount of shrinkage, the longer the cycle, and the higher the cost. Of course the opposite of this is also true. In fact, under certain molding conditions, once the part is out of the mold it may continue to cool and shrink for up to 30 days. Admittedly, the first 95% of the cooling and shrinking takes place within the first few minutes after removal from the mold, but that last 5% can take up to a month to stabilize and finalize. Even if the shrinkage is controlled to achieve that first 95% through molding parameter adjustments, the theoretical cycle time could evolve into 10 minutes for a part that we know should normally run at a 30-second cycle. One way of minimizing the cycle while controlling the shrinkage is to control the shrinkage after the product is ejected from the mold instead of while it is still in the mold. The cycle time can be reduced, thus the cost of molding can be reduced. This is what post-mold cooling and shrinking is all about.

Post-mold shrinkage is normally controlled by restraining the molded product in a fixture that holds it in place while it cools. An example is shown in Figure 1-18.

Notice that the product is being purposefully bent and bowed in directions opposite the normal shrinking and cooling patterns that develop when a part cools. This is to overcompensate for shrinkage so the part will spring back to a desired

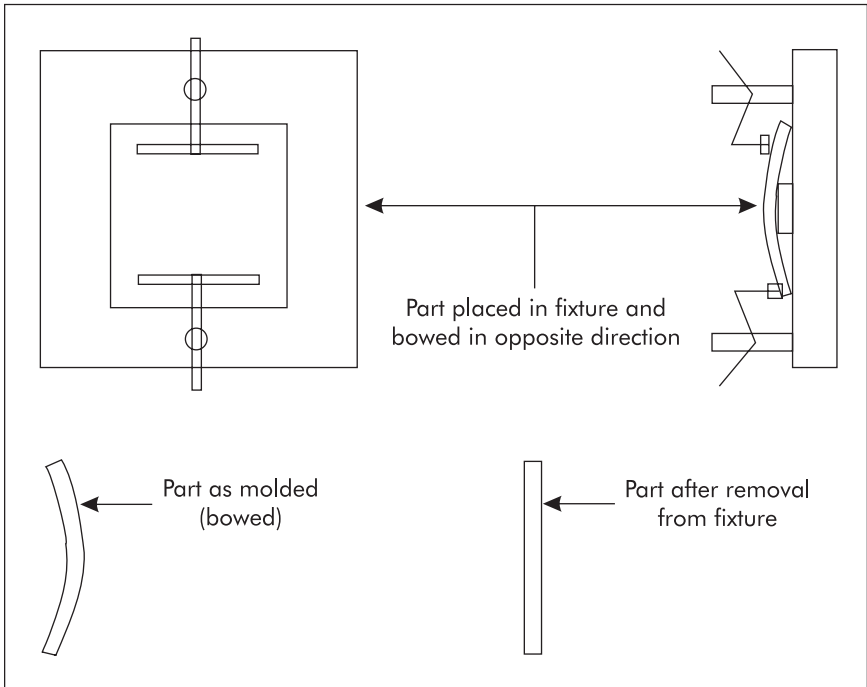


Figure 1-18. Post-mold shrinkage fixture.

shape after cooling. This must be done through trial and error by measuring cooled parts to determine how to adjust the fixture to give the desired results.

When using post-mold cooling/shrink fixtures, it is necessary to leave the cooling product in the fixture for the equivalent time of approximately six full cycles. Therefore, it is necessary to have at least six fixtures, or “stations” in place at all times. Forcing air over the parts helps stabilize them.

Another method of post-mold cooling is to simply drop the molded parts in a container of cold water. The temperature of the water must be maintained at below room temperature (approximately 60° F [16° C]), but there is no advantage in having it lower than that because once the entire plastic mass drops to a temperature below its melting point or glass transition point, it will not continue to shrink. The post-mold cooling is only being done to effect cooling for the center portions of the walls that take longer to solidify than the external skin of the walls.

There is a danger in using any method of post-mold shrinkage control because the practice does induce varying degrees of mechanical stress to the molded product. This stress is caused by the forcing of molecules into positions that they are not seeking on their own. When this is done, stress is concentrated on

the molecules that are being stretched and compressed as will be discussed in the following section. This stress is maintained as the part cools and is locked in after the part has fully cooled and shrunk. Then, if it is ever exposed to extreme temperatures or mechanical abuse, the stress is relieved and the product may fracture, crack, or shatter, depending on how much stress was induced during the post-mold shrinkage control.

MINIMIZING MOLDED-IN STRESS

Other than contamination, the single most significant cause of field failure of an injection-molded product is molded-in stress. Stress can be defined as: *resistance to deformation from an applied force*.

All this means is that if a force is applied to an object, the object resists having its shape changed, as depicted in Figure 1-19. The amount of resistance that is present can be identified as stress.

It is possible to understand molded-in stress by visualizing what happens during the injection molding process. A plastic material is heated to a temperature at which it assumes the consistency of warm honey and is ready to inject into a

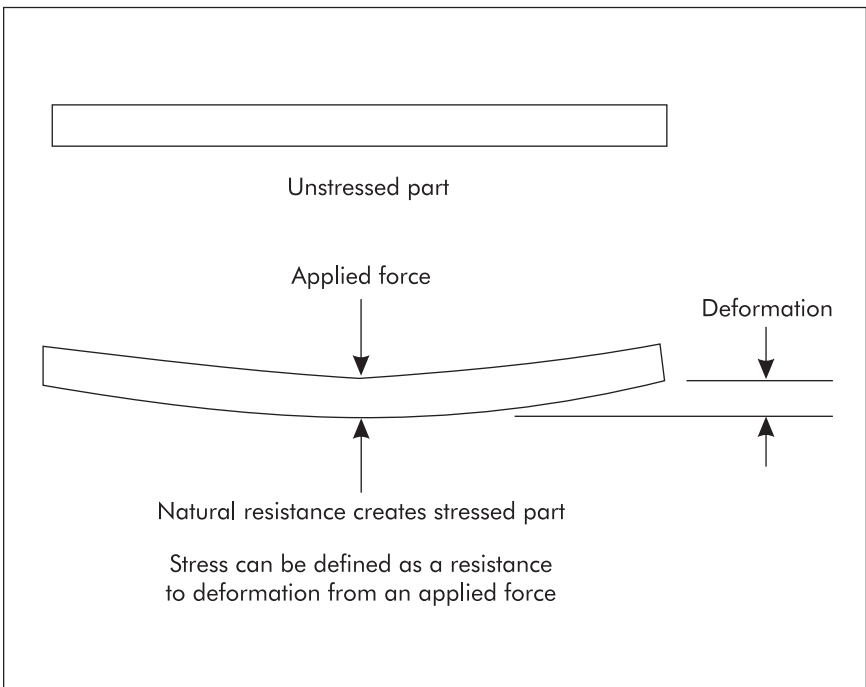


Figure 1-19. Defining stress.

mold. During this heating phase the molecules of the plastic begin to move around. This is what actually causes the material to soften, or melt. Once the material is at the proper temperature, a plunger device (screw) injects the material by pushing it forward through the machine and into the mold. This pushing action causes the molecules to align, or orient, in a linear fashion. It is similar to pushing a fork through a plate of cooked spaghetti; the pasta strands (molecular chains) start to line up neatly next to each other in the direction that the fork is traveling.

In the molding process, these molecular chains are injected into a mold where they are then cooled while still being held under high pressure. Because they are kept from relaxing and reverting to their original state, they are solidified under stress. It's like stretching a rubber band, and then freezing (solidifying) it in that stretched-out position. If the rubber band thaws, it will snap back towards its original state. That happens because stress is being released. The same thing happens in an injection-molded part. If the part is allowed to relax after solidifying, either by exposure to elevated end use temperatures or being knocked sharply against the edge of a desk, the stresses that were molded in are released, and warpage, cracking, twisting, crazing, or even shattering can occur.

To minimize the possibility of any of these things occurring in the molded part, it is best to minimize the amount of stress that is molded in. Although it may not be possible to eliminate all of it, it is possible to minimize at least a portion of it. This can be done through product design as well as mold design, proper material selection, and control of processing by adjusting and maintaining molding parameters.

SUMMARY

The injection molding industry started in 1868 with the invention of an injection molding machine and process by John and Isaiah Hyatt. It did not flourish until the 1940s with the advent of World War II, during which the government sought new, fast, inexpensive methods and materials for producing products of all kinds.

Injection molding can be defined as a process that consists of heating a plastic material to a point at which it becomes soft enough to force into a closed mold, at which point the material cools enough to solidify and form a specific product.

There are more than 100 parameters in the molding process that must be controlled to ensure the proper manufacture of a product. All fall into one of four critical categories: time, temperature, pressure, and distance.

The softening (or melting) of the plastic is achieved by causing the individual molecules within the plastic material to go into motion. This is accomplished by applying heat.

There are four heating zones in a molding machine heating cylinder: nozzle, front, center, and rear.

Injection pressure is used to fill the mold to its primary capacity (95%) with molten plastic. Clamp pressure is used to keep the mold closed against the injection pressure of the incoming molten plastic.

Total cycle time of the injection molding process consists of totaling the individual times required for a variety of activities, including (among others) injection time, cooling time, and machine closing and opening times.

Insert molding and overmolding (a form of insert molding) are usually performed on a vertical molding press. This allows loading and unloading to be performed while another set of parts is being molded, thus reducing operating costs substantially.

Material property values can be altered significantly by adjusting molding parameters.

Setup sheets are vital to successful operation of a molding machine in production. However, it is important to understand that a molding process will take up to 8 hours to stabilize and a setup sheet should contain data for both the start-up phase and subsequent stabilized phase of the process.

All materials have a specific “shrinkage rate” value assigned to them by the material manufacturer. The shrinkage rate is a value that can be used to predict how much difference there will be between the plastic product when it is first molded and after it has cooled to room temperature.

QUESTIONS

1. During what year did John Wesley Hyatt start the injection molding industry by winning a contest for inventing the process?
2. In what year did John and Isaiah Hyatt receive a patent for the first injection molding machine?
3. Why were the 1940s instrumental in creating an explosion of interest and growth in the plastic injection molding industry?
4. During what year did plastic production surpass steel production in the United States?
5. Describe the injection molding process as it is defined in this chapter.
6. How many parameters must be controlled during the injection molding process?
7. How many categories are required to list all the parameters noted in question no. 6?
8. What is the recommended melt temperature for liquid crystal polymer material?
9. What causes the plastic material to melt (or soften)?
10. Not counting the nozzle, name the heating zones present in the injection heating cylinder.

11. What is the recommended mold temperature for liquid crystal polymer material?
12. What device is used to control the temperature of the hydraulic oil in a molding machine?
13. What are the three primary types of pressure found in the injection molding process?
14. What is the reason for wishing to obtain a melt index value for a specific material?
15. Define the term “clamp pressure.”

The Language of Plastics

DEFINITIONS RELATING TO MATERIALS

In this chapter we provide definitions for the various terms that are unique to plastic materials. These descriptions are important because they help in the decision-making process for selecting the proper material or family of materials to provide the required properties for a specific product and process. We start with the most pervasive term, *plastic*.

The Definition of Plastic

A simplistic definition of plastic (as used to describe molding materials) might be:

Any complex, organic, polymerized compound capable of being shaped or formed.

Generally speaking, the terms “plastic” and “polymer” are used interchangeably, although strictly speaking a polymer is a plastic, but a plastic does not have to be a polymer. Plastics can be in the form of liquids or solids or something between the two.

Plastics are created by refining common petroleum products, crude oil and natural gas being the main building blocks. Figure 2-1 is a diagram of how these blocks are utilized in making some of the more common plastic materials available today. Experimental work is currently underway to create plastic materials from sources other than petroleum, with limited success recorded in creating primary materials from such products as vegetable oils and coal.

Polymerization

When we discuss *plastics* we are usually referring to compounds that have been created by way of a process called *polymerization*, defined as:

A reaction caused by combining monomer with a catalyst, under pressure, and with heat.

A *monomer* is a single unit. In the polymerization process we combine many single units of plastic into many combined units of plastic, known as *polymers*. Therefore, the process used to produce these polymers is called *polymerization*. The polymers are what we use for molding. Polymers are formed by combining a series of monomers. We will take a look at how this is accomplished.

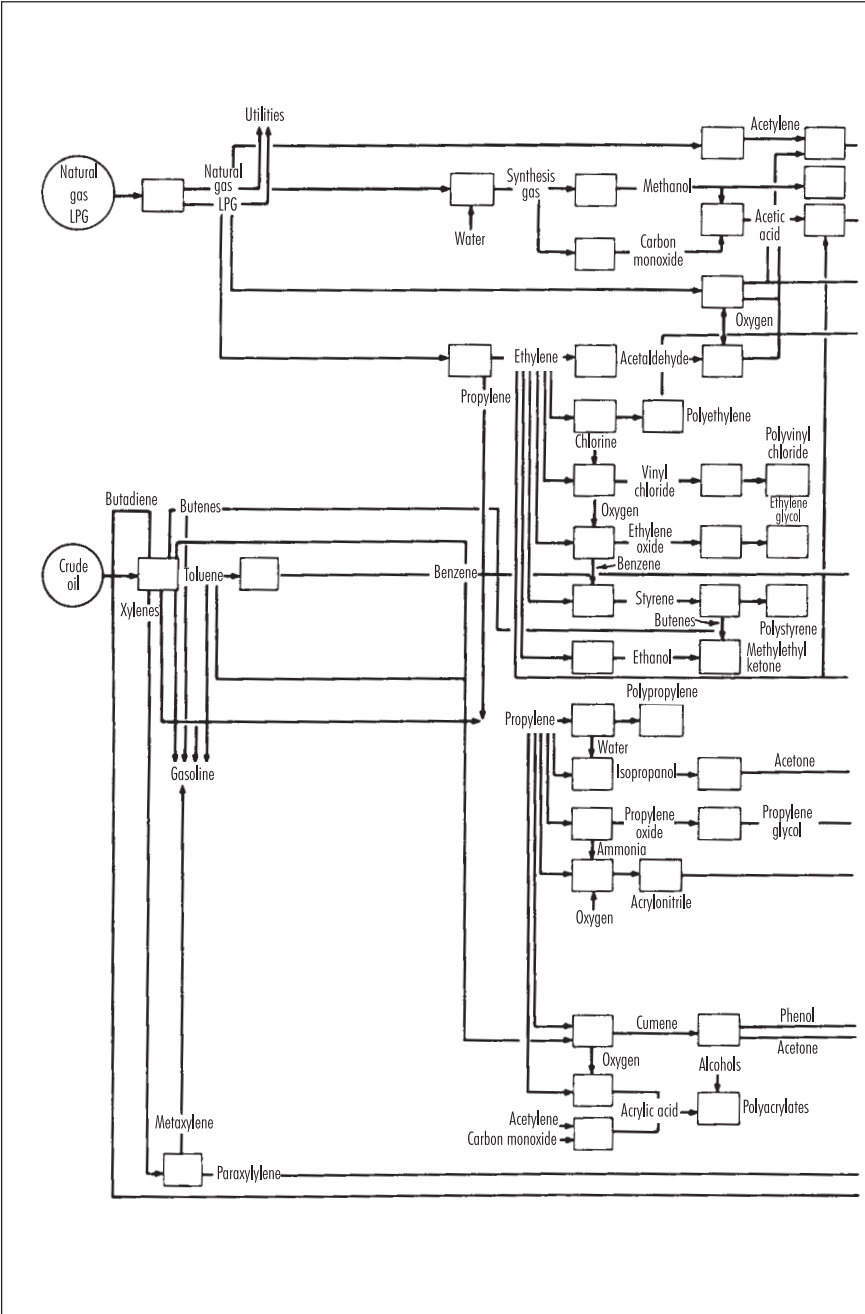
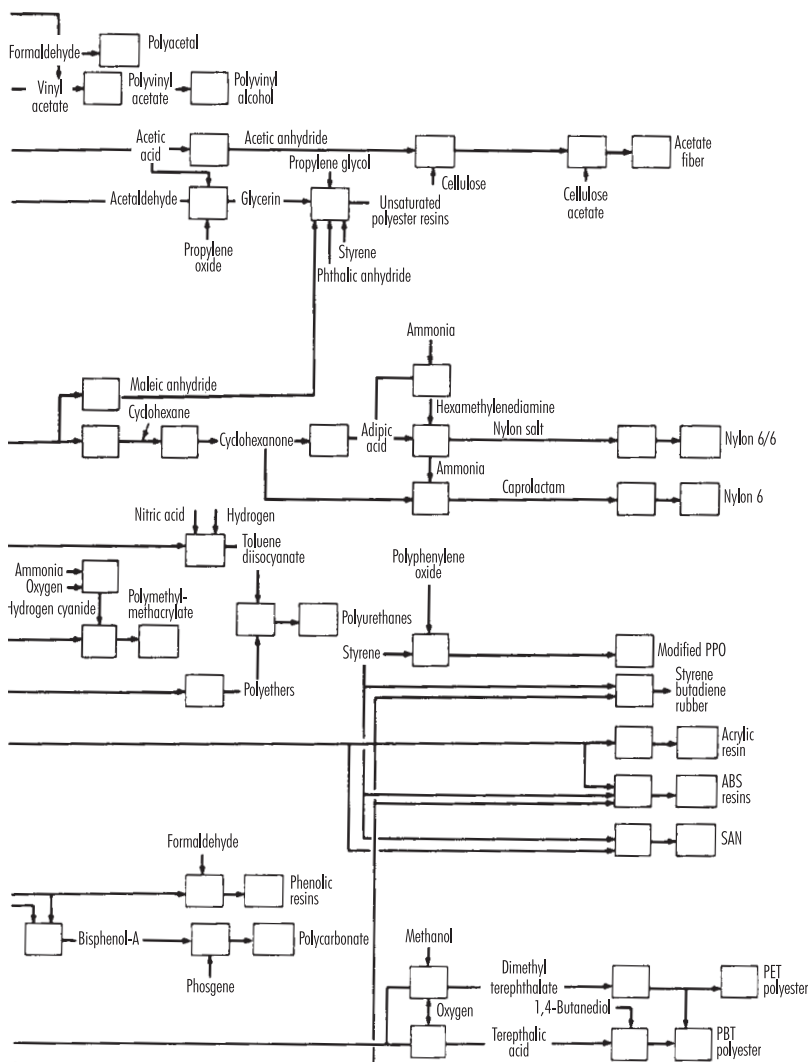


Figure 2-1. Building blocks for common plastic materials are derived from petroleum



products, refined to provide specific properties. (Courtesy Celanese Plastics Co.)

Basic Molecular Structure

Assume that you owned a very powerful microscope capable of seeing molecules. If you are viewing a plastic molecule, you would see only a whirling ball of motion. This would be a bunch of elemental atoms spinning and vibrating around a central location. If you could invoke a “stop-motion” action on the microscope, you would see that these atoms appear to be held together, but there would be no strings or lines connecting them. They seemingly are held together by some invisible force of attraction. We represent this phenomenon graphically by drawing atoms connected by lines, as shown in Figure 2-2, an example of an ethylene monomer molecule.

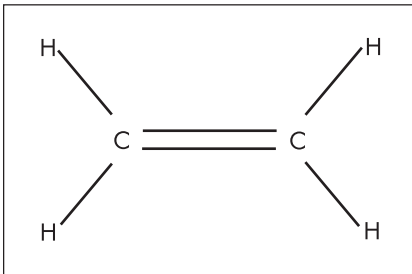


Figure 2-2. An ethylene monomer.

Atoms in Figure 2-2 are represented by the element identifiers, H for hydrogen and C for carbon. We show lines to represent the forces that connect the atoms from all directions, but in reality there are no actual visible connections. The double lines between the carbon elements represent an area of the molecule that will be utilized to form bonding sites during the polymerization process. When a polymer is formed, it is done so by using a process that temporarily breaks down a series of monomer molecules and then connects them in

one large polymer molecule formed by linking the monomer molecules at specific “bonding sites.” Prior to this polymer formation, the multiple monomer molecules look like those shown in Figure 2-3.

Note that all the ethylene monomers look the same, but they are not connected to each other. Think of them as loose railroad cars, all on different tracks in a train yard. To become a polymer of ethylene, the monomers must be connected to each other in a specific way. This is accomplished by exposing them to the polymerization process. When the polymer has formed, the line representation looks like that shown in Figure 2-4.

All the monomers now have been connected to form a polymer of ethylene. This is known as a plastic called *polyethylene*. Note that the double lines between the carbon elements have been separated into single lines found on both sides of the carbon elements. These are the bonding sites. The result is a chain of ethylene molecule units tied together to form the single ethylene polymer. In the case of polyethylene, approximately 1,000 monomer units are tied together to form a single polymer unit. Laid end to end, it would take approximately 100,000 of these polymer units to equal 1 in. (25.4 mm) of length.

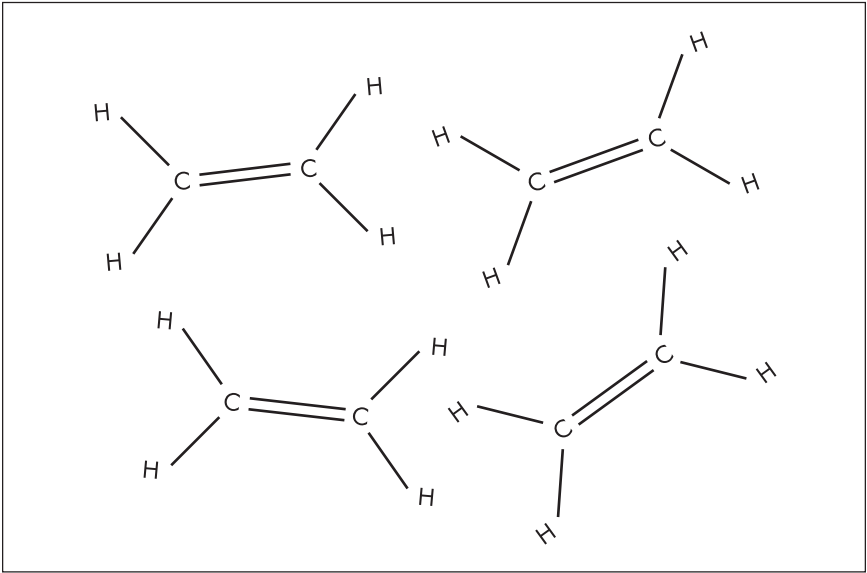


Figure 2-3. Multiple ethylene monomers prior to polymerization.

So, while a monomer is a single unit of plastic, a polymer is many units of plastic connected together. This is accomplished through the polymerization process which mixes monomers with a catalyst and adds pressure and heat to complete the connecting process.

Copolymers

To create copolymers, it is necessary to react monomers of different groups together. For example, a monomer of styrene can be reacted with a monomer of acrylonitrile and a monomer of butadiene to form the familiar ABS (acrylonitrile-butadiene-styrene) copolymer shown in Figure 2-5.

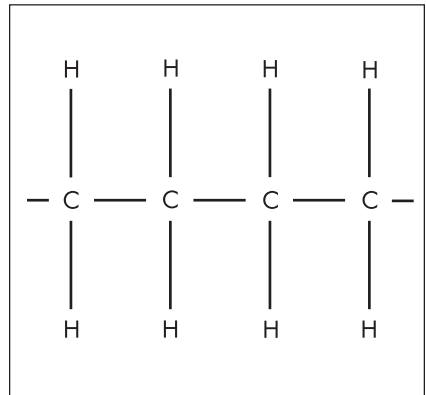


Figure 2-4. POLYmerized ETHYLENE monomers (Polyethylene).

Altering the amount of each monomer will create totally different property values. Copolymers can be produced using any of several polymerization techniques (these techniques are discussed later in this chapter under “How Plastics Are Made”).

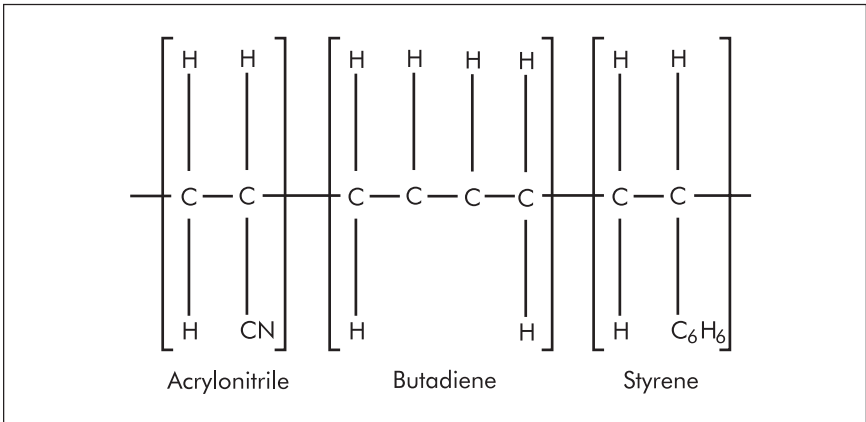


Figure 2-5. Copolymer molecular structure (ABS).

Alloys and Blends

In addition to polymers and copolymers, many basic plastic formulations also are available as alloys and blends. An alloy is a material formed by mixing two (or more) basic plastics to form a “new” plastic. This new plastic is a homogenous mixture of the basic plastics, usually exhibiting improved properties over the original plastics. A blend, on the other hand, is also a mixture of two (or more) basic plastics, but the resultant mixture is not homogenous and merely exhibits an average of the property values of the original plastics. When discussing alloys and blends with material suppliers it is important to understand the difference and to make sure the distinctions of both are understood.

THERMOPLASTICS VERSUS THERMOSETS

While there may be many different ways to classify the approximately 20,000 plastic materials available today, the first classification is usually the one that defines the material as either thermoplastic or thermoset.

Thermoplastic materials can be defined as:

A plastic material which, when heated, undergoes a PHYSICAL change. It can be reheated, and reformed, over and over again.

Examples of thermoplastic materials are ABS (used for such applications as TV cabinets and computer housings) and polyethylene (used for milk jugs, food wrap, etc.).

Thermoset materials can be defined as:

A plastic material, which when heated, undergoes a CHEMICAL change and “cures.” It cannot be reformed, and reheating only degrades it.

Examples of thermoset materials include phenolic (used for automotive distributor caps, for example) and melamine (used for plastic dinner plates).

There is a common perception that thermoset materials cannot be recycled because they “cure” when processed. However, curing only means that they cannot be melted down again, as can thermoplastics. Molded thermosets *can* be ground into a powder and used as a filler in making other thermoset compounds or even thermoplastic compounds. They should never simply be discarded to a landfill.

The focus of this book and other volumes in this series is thermoplastic materials because of their current popularity (approximately 80% of all plastic products today are made from thermoplastic materials). Thermoplastics are available in a wide variety of primary formulations as well as many blends and alloys. They are easily processed and simply require heating, forming in a mold, and cooling. They can be compared to water in the sense that when heated they can change from a solid to a liquid, and back again when cooled, over and over, without altering their chemical makeup. But unlike water, they are identified in one or the other of two distinct categories: amorphous or crystalline.

Amorphous versus Crystalline

The two major categories of thermoplastic materials are amorphous and crystalline. While some materials are available in either category, and some materials are actually combinations of both, in our discussion we classify them as either amorphous or crystalline. There are some major differences between the two.

Amorphous Materials

Amorphous (am-OR-fuss) materials are those in which the molecular chain structure is random (as shown in Figure 2-6) and becomes mobile over a wide temperature range. This simply means that these materials do not literally *melt*, but rather *soften*, and they begin to soften as soon as heat is applied to them. They get softer and softer as heat is absorbed, until they degrade as a result of absorbing excessive heat. However, it is common and acceptable to refer to amorphous materials as “melting,” and we do so in our discussion.

Crystalline Materials

In crystalline (CRISS-tull-in) materials, the molecular chain structure is well ordered (as shown in Figure 2-7) and becomes mobile only after the

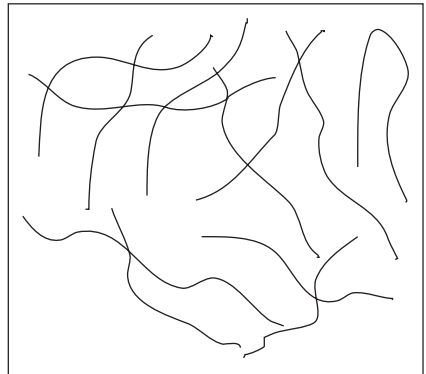


Figure 2-6. Amorphous molecular chains.

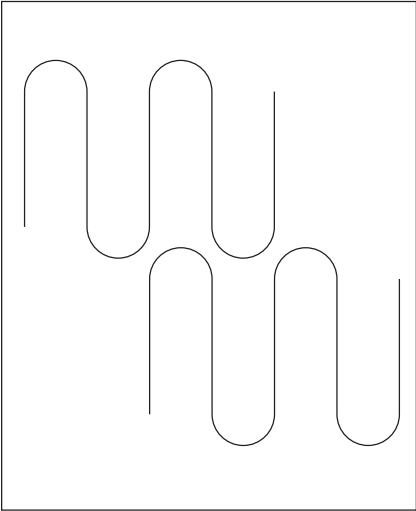


Figure 2-7. Crystalline molecular chains.

material is heated to its melting point. That means these materials do not go through a softening stage but stay rigid until they are heated to the specific point at which they immediately melt. They will degrade if excessive heat is absorbed.

Comparison of Amorphous and Crystalline Molecular Chains

Owing to the molecular structure of these two types of materials, their physical properties are worlds apart. In fact, they are just about directly opposite each other. For example, products molded from these materials would have the properties shown in Table II-1.

Table II-1. Amorphous versus Crystalline Properties

Amorphous	Crystalline
<ul style="list-style-type: none">• Clear• Low shrinkage• Softens (no melt)• High impact• Poor chemical resistance• Poor lubricity	<ul style="list-style-type: none">• Opaque• High shrinkage• Melts (no soften)• Low impact• Good chemical resistance• Good lubricity

However, for every rule there is an exception, and this holds true with plastics also. For instance, even though ABS is an amorphous material, it is *not* clear, but translucent. In general, however, the comparisons stated hold true. In addition, some materials (such as nylon) are available in either amorphous or crystalline formulations, and there are some materials that are semicrystalline, which have both amorphous and crystalline properties.

Table II-2 shows the classifications of some of the more common plastics.

Elastomers

Elastomers are polymers that exhibit a wide spectrum of elastic or rubber-like properties. Typical elastomeric materials are shown in Table II-3.

Table II-2. Examples of Amorphous and Crystalline Materials

Amorphous Materials	Crystalline Materials
<ul style="list-style-type: none">• ABS• Acrylic• Cellulose propionate• Polyamide-imide• Polyarylate• Polycarbonate• Polyetherimide• Polyethersulfone• Polyphenylene oxide• Polystyrene• Polyurethane	<ul style="list-style-type: none">• Acetal• Cellulose butyrate• Liquid crystal polymer (LCP)• Nylon• Polyester (PBT)• Polyetheretherketone (PEEK)• Polyethylene• Polyethyleneterephthalate (PET)• Polyphenylene sulfide• Polypropylene• PVC

Table II-3. Typical Elastomers

<ul style="list-style-type: none">• Acrylates• Fluorocarbons• Polyurethanes• Silicones	<ul style="list-style-type: none">• Butyls• Fluorosilicones• Neoprenes• Styrene butadienes	<ul style="list-style-type: none">• Chlorosulfonated polyethylene• Polysulfides• Nitriles
---	---	---

HOW PLASTICS ARE MADE

We’ve explained that plastic molding materials are created through polymerization. Within the process itself are three common ways of polymerizing materials. The first is called *addition*-type polymerization in which a simple combining of molecules occurs without the generation of byproducts. Actually, addition- type polymerization can be achieved in several different ways, but the most common involves chemical activation of the molecules, which causes them to combine with each other in a chain reaction. This method is used for creating ethylenes.

A second type of polymerization, called *condensation*, involves removing specific atoms from each molecule, thereby allowing the molecules to link together. In this process certain byproducts are created which must be removed from the reacting polymers so that further polymerization is not inhibited. This method is used for making some nylons and phenolics.

Combining the addition and condensation polymerization process gives us the third type, *combination*.

In this method, a polymer is first formed using the condensation method. Then this still active polymer is exposed to an additional polymerization process which causes further reaction to form larger polymers with a third ingredient. This is the process by which “cross-linking” polyesters are made.

MOLECULAR WEIGHT AND DISTRIBUTION

The physical properties of any given polymer are controlled by a variety of factors. The most important are generally considered to be the polymer’s molecular weight (MW) and its molecular weight distribution (MWD).

Molecular weight is the average weight of the molecules in the mixture that makes up the polymer. Polymers with higher average molecular weight tend to be tougher and more chemically resistant, while those with lower average molecular weight tend to be brittle and weak. However, the higher the molecular weight, the more difficult it is to process the material during molding operations.

Molecular weight distribution is defined as the relative proportions of molecules of different weight within a polymer. It is a measurement of the ratio of large, medium, and small molecular chain lengths found in the material. If the material is made up of molecular chains that are mostly near the average length, it is called narrow-range. If it is made up mostly of chains of a much wider variety of lengths, it is called broad-range. Generally speaking, the narrow-range polymers have better mechanical properties than the broad-range polymers, but they are much more difficult to process.

INFLUENCE OF TIME AND TEMPERATURE

The actual molecular structure of any given polymer can be altered by exposing it to varying time and temperature processes. For example, heating a crystalline polymer above its melting point and then quickly quenching it will create a material that is more amorphous in nature. This is because quenching interferes with the ability of the molecules to reform normal crystals found in crystalline materials. Thus, the quenched product will exhibit properties more closely related to amorphous materials than crystalline materials. Slower cooling will enable the crystals to form, resulting in a product with properties associated with crystalline materials.

Given enough time, heated polymer molecules will migrate to find their ideal *preferred* equilibrium structure. Increasing the temperature allows the molecules to accomplish this equilibrium in a much shorter time, but reducing the

temperature minimizes the degree of equilibrium achieved. Thus, the property values of a polymer may vary widely, depending on how much time was allowed for the molecular structure to equilibrate.

SUMMARY

A simple definition for plastic, as it is used in this book, is: any complex, organic, polymerized compound capable of being shaped or formed. Usually, the terms “plastic” and “polymer” are used interchangeably, although strictly speaking a polymer is a plastic but a plastic does not have to be a polymer.

Plastics are created by refining common petroleum products. Crude oil and natural gas are the main building blocks for plastic materials.

Polymerization can be defined as a reaction caused by combining monomers with a catalyst, under pressure, and with heat applied.

A monomer is a single molecular unit, while a polymer is made up of combined monomers.

Many basic plastic materials are also available as alloys and blends. An alloy is a material formed by mixing two (or more) basic plastics to form a “new” plastic that is totally homogenous, while a blend is simply two (or more) plastics mixed together without becoming homogenous.

Thermoplastic materials are those which, when heated, undergo a physical change and can be reheated and reshaped over and over again.

Thermoset materials are those which, when heated, undergo a chemical change and actually become something new. They cannot be reformed by being reheated.

Amorphous materials are those in which the molecular chain structure is random and becomes mobile over a wide temperature range. Crystalline materials are those in which the molecular chain structure is very orderly and becomes mobile only after applying enough heat to reach its melting point.

Plastics are polymerized using one of three basic methods referred to as *addition*, *condensation*, and *combination*.

The most important factors in controlling the physical properties of any polymer are molecular weight and molecular weight distribution.

Given enough time, heated polymer molecules will migrate to find their ideal “preferred” equilibrium structure. Increasing the temperature allows the molecules to accomplish this equilibrium in much shorter time, but reducing the temperature minimizes the amount of equilibrium achieved. Thus the property values of a specific polymer may vary widely depending on how much time was allowed for the molecular structure to equilibrate.

QUESTIONS

1. What is the definition of “plastic” as it is used in this book?
2. Define the term “polymerization.”
3. What naturally-occurring products are the main building blocks of plastics?
4. What is necessary to create copolymers?
5. What is the major difference between an alloy and a blend?
6. Define “thermoplastic.”
7. Define “thermoset.”
8. What are the two major classifications of thermoplastic materials?
9. Define “amorphous” materials.
10. Define “crystalline” materials.
11. List three properties of typical amorphous materials.
12. List three properties of typical crystalline materials.

Reinforcements and Fillers

WHY REINFORCEMENTS AND FILLERS ARE USED

Most plastics today are available in forms that are beyond their “pure” state. The process of adding chemicals and organic and inorganic materials has been developed to improve one or more of the properties of the pure polymer. For instance, small amounts of lubricant in the form of fatty esters may be added to improve the flow properties of a given plastic.

Additives usually fall into two basic categories: reinforcements and fillers. While it is true that a reinforcement may be considered a filler, it is not necessarily true that a filler can be considered a reinforcement. Reinforcements are those additives that are used to enhance physical strength properties. Fillers are additives used to enhance properties other than strength.

WHAT ARE THEY?

As we take a look at some common additives in this section, we will see what they are made of and how they affect basic polymer properties. To begin, we look at reinforcements. These are used to enhance physical strength properties.

Reinforcements

Fiberglass

Fiberglass is the most popular reinforcement additive today. Sometimes referred to as glass fiber or glass filler, it takes many forms, the most common being short, individual strands (*chopped roving*) approximately 1/8 to 1/4 in. (3.2 to 6.4 mm) long and usually less than 0.001 in. (0.03 mm) in diameter. Another popular form is in the shape of spheroids (balls) called *hollow spheres*, with an average diameter of approximately 0.003 in. (0.08 mm). Fiberglass is also available in *long strand* (3/8 in. [9.5 mm] or greater) and in *flake* form, although these are very difficult to process in injection molding operations and are not frequently utilized.

Normal loadings of fiberglass are in the 30% range (by weight). In some cases this can be as high as 80%, but processing is extremely difficult at these levels. Even a level as low as 5% can greatly increase physical properties.

Incorporating glass fibers into polymers affects all physical properties, especially tensile and flexural strengths, heat deflection temperature, impact strength, and elongation. Table III-1 shows how these properties are affected for a specific polycarbonate application. The amount of glass reinforcement is 20% by weight.

Table III-1. Effects of 20% Glass Reinforcement on Polycarbonate

Nonreinforced				
Flexural Strength $\text{psi} \times 10^3$	Tensile Strength $\text{psi} \times 10^3$	Impact Strength IZOD ft-lb/in.	Specific Gravity	Thermal Coefficient of Expansion $\text{in./in.}^\circ \text{F} \times 10^{-6}$
13	9	16	1.20	39
Reinforced (20% glass)				
Flexural Strength $\text{psi} \times 10^3$	Tensile Strength $\text{psi} \times 10^3$	Impact Strength IZOD ft-lb/in.	Specific Gravity	Thermal Coefficient of Expansion $\text{in./in.}^\circ \text{F} \times 10^{-6}$
17	12	1.5	1.34	12

There are two items worth noting in Table III-1. First is the specific gravity value. This value for reinforced plastic is higher, which means the molded product will weigh more than if the material is not reinforced. Therefore, not only does the raw material cost more per pound because reinforcement is added, but there is an additional cost to absorb because more material (by weight) will be used to mold a specific product.

The second item to consider from the table is the IZOD impact strength. Due to the orientation of the fiberglass strands during molding, the physical strength is increased in certain directions. The IZOD impact test actually tests the impact strength at right angles to this orientation, which results in weaker readings for the reinforced material versus the nonreinforced material. Therefore, a reduction in IZOD impact test results should be expected.

In addition to the effects shown in Table III-1, is another important effect: use of glass reinforcement will enhance the electrical properties of any plastic material. Chemical resistance is also increased, as is heat resistance.

Glass reinforcement is available to the molder in three basic configurations. The most popular version is that in which glass is mixed with the basic polymer at time of manufacture by a compounder. The result is usually a pellet shaped like a cylinder 1/8 in. (3.2 mm) in diameter by 1/8 in. long. The glass that is

added is an integral part of that pellet, so each pellet contains the appropriate percentage of glass.

The second most popular version is that in which special polymer pellets are compounded with 60 to 80% glass concentration. The pellets are then added by the molder to the machine hopper containing the standard, unreinforced polymer pellets. By altering the quantity of concentrated pellets, the glass percentage of the final melt can be adjusted to whatever level (percentage) required.

The third version is that in which the molder purchases chopped strand glass and adds it directly to the polymer in a *pre-blend* operation. This blend is then fed to the hopper of the injection molding machine. This process does require special handling equipment, however, because of the shape and condition of the pellets.

Organic Fibers

Organic fibers are used when lighter weight, more stiffness, and higher tensile strength than obtainable from glass are required. These fibers are also radar and sonar transparent. Their lower specific gravity (approximately 50% that of glass) helps reduce the weight. Significantly, though organic fibers have these outstanding properties, they are processed as easily as glass. And, although they are more expensive than glass on a per-pound basis, their lower specific gravity usually makes them equal to glass on a per-in.³ (volume) basis.

The most popular organic fiber today is an *aramid*. The term aramid refers to a special class of *aromatic polyamide* fibers. This is a synthetic material first developed by the DuPont company under their tradename, *Kevlar*TM. Aramids are available in many standard forms (fiber, roving, fabric, etc.) and are available as hybrids combined with other reinforcement materials such as glass and graphite.

Boron

Due to its high cost per pound (as much as \$100 in January of 1996) boron is not typically used as a reinforcement for injection mold. However, boron filaments possess a combination of strength, stiffness, and density not equaled by any other continuous filament form of reinforcement material. It is used primarily for high-strength applications such as fabrication of stabilizer sections for F-14 fighter aircraft.

Boron fibers are usually produced in filament form by depositing the material on a substrate filament (0.0005 in. [.001 mm] diameter) of tungsten. The finished product has a high specific gravity (slightly higher than glass), but the resulting physical properties make it an option for use in structural applications such as aircraft components and space platform projects, as well as for sporting goods such as golf club shafts and tennis racket frames. Boron is often combined with other reinforcement materials such as aramid, glass, and carbon-graphite.

Carbon-graphite

Low cost (under \$10 per pound) and excellent modulus-to-weight and strength-to-weight ratios make carbon-graphite fibers ideal substitutes for glass and metals. Because of their low thermal expansion, these fibers can be used to minimize mold shrinkage and can even create materials with zero mold shrinkage.

Carbon fibers have been in existence for more than 100 years. Thomas Edison used a filament of carbonized sewing thread to make the first commercial glass lighting lamp in 1879. Among the excellent property values of carbon-graphite fibers are high modulus and strength, low coefficient of expansion, low coefficient of friction, low density, and resistance to most chemical exposure. In addition, carbon-graphite fibers are used to make plastic materials (which are inherently insulators) electrically conductive.

A medium-high specific gravity of 1.6 to 1.9 g/cm³ makes this material an excellent choice for cost-effective reinforcement fibers. They can be combined with other reinforcements, such as glass and boron, for even greater property values. Typical applications include aircraft components, fishing rods, tennis rackets, racing bicycles, skis, canoes, and bows and arrows.

Ceramics (Metal Oxides)

Ceramic fibers are continuous strands of metal oxides that exhibit extremely high temperature resistance (3000° F [1649° C]), as well as high modulus and excellent compressive strength. Ceramic fibers are typically produced using aluminum, silica, quartz, and zirconia.

As might be expected, these fibers have high specific gravity values of 2.5 g/cm³ and greater, and are priced in the range of \$15 per pound or higher. They have low tensile strength properties, however, and this must be considered during the product design stage. A wide range of product applications exists, from water filtration systems to jet engine blades and bearings. Frequently, these materials are used to produce common forms such as rods, tubes, and billets, which are then machined to final shape.

Metals

Metals used as reinforcement fibers for plastics include such materials as stainless steel, steel wire wool, nickel flakes or strands, titanium, silver, gold, and platinum. While some of these are rarely used, others (such as nickel and stainless steel) are used frequently. Metals provide unique physical and thermal properties, and also feature very controlled fiber geometry.

Metal fiber costs vary greatly depending on the application, fiber diameter and length, and the cost of the base metal itself. The less expensive metals can cost in the area of \$2 per pound, while at the high end these materials can cost as much as \$50 per pound. Of course, the more exotic metals such as gold and

platinum can cost 10 times that much. Table III-2 shows some of the distinguishing characteristics of various metal fibers.

Besides offering standard metallic properties to plastics, these fibers also offer electromagnetic interference (EMI) shielding to plastic components used to house electronic products. Although tests have shown them to be less effective than metallic paints or plating, the fibers can be used in less stringent designs where molded-in EMI shielding is permissible. This results in a low-cost method of EMI shielding requiring no secondary operation or special surface preparation as with the other methods.

Table III-2. Metal Fiber Characteristics

Metal or Alloy	Outstanding Characteristics
• Carbon and alloy steels	High strength, high modulus, low cost
• Stainless Steel	Good corrosion resistance
• Nickel and cobalt	Resistance to elevated temperatures, oxidation resistance
• Refractory metals (Ti, Ta, Nb)	High melting points, good corrosion resistance
• Copper and copper alloys	High electrical and thermal conductivity
• Aluminum and aluminum alloys	Low density, low cost, electrical/thermal conductivity
• Precious metals (Pt, Ag, Au)	Electrical/thermal conductivity, corrosion resistance

From *Handbook of Reinforcements and Fillers for Plastics*, Katz and Milewski, Van Nostrand Reinhold.

Fillers

Fillers are used to enhance properties—other than structural—of plastic materials. They usually add cost to the plastic but, as in the case of clay, may actually reduce the cost of the plastic itself. In most cases, however, the addition of fillers will interfere with the processing of the plastic and the physical property values. This interference is exhibited at various levels depending on the filler used and the percentage of filler versus the percentage of plastic. Addition of filler normally results in the need to adjust temperatures and pressures upward. Also, be aware that most additives will absorb moisture, which may add to the drying requirement for a specific plastic.

Fillers are usually added to the plastic so that they comprise from 10 to 50%, by weight, of the resultant compound. Table III-3 shows some common fillers and the reason(s) for using them.

Other fillers are used for more specific reasons. Discussion of some of the more common ones follow.

Table III-3. Common Fillers and Their Uses

Filler	Reason for Use
<ul style="list-style-type: none">• Wood flour, sawdust, wood pulp, sisal-jute, purified cellulose, mica-rock	Add bulk to the plastic, reduce cost
<ul style="list-style-type: none">• Inorganic pigments, mineral-powders, metallic oxides, powdered metals, graphite, silica	Increase hardness
<ul style="list-style-type: none">• Diatomaceous earth, ceramic oxides, silica	Increase thermal resistance
<ul style="list-style-type: none">• Glass fibers/fabrics, synthetic fibers/fabrics, graphite, metallic oxides	Increase chemical resistance

Colorants

The use of *colorants* allows a wide variety of plastics to be available in colors ranging from mild pastels to vibrant deep hues. When using a plunger injection machine, you can even attain a marbled, or varicolored, effect. (A screw injection machine creates a homogenous blend that does not allow marbling.)

Generally speaking, two types of colorants are used for plastics: dyes and pigments. Both are available as organic and inorganic compounds. The major difference between dyes and pigments is their respective degree of solubility. Dyes tend to be soluble in plastics, while pigments, being insoluble, are simply well dispersed throughout the plastic mass. The decision of which to use is based on the level of resin compatibility needed and the need for solubility. The next step is to determine the level of need for color stability. This is the ability of the colorant to maintain the proper color after exposure to processing techniques as well as end use environmental exposures (including air, water, and light).

Although dyes are soluble, they are subject to color migration, or bleeding. This may not cause a problem with the physical structure of a molded product, but it may have aesthetic ramifications for products such as jewelry.

The use of pigments results in plastics that are opaque and light-sensitive, with a fading on the surface and obscured pigment beneath. Undispersed pigment particles may negatively affect the physical properties of the plastic.

Plasticizers

Flexibility in a plastic is achieved by using *plasticizers* during initial compound-ing. Hundreds of plasticizers are available, and although polyvinyl chloride (PVC) consumes the largest amount of available plasticizers (80%), other plastics, including acrylonitrile-butadiene-styrene (ABS), polystyrene, and nylon also have minimal requirements for flexible versions. Plasticizers act as internal lubricants

in that they overcome the natural attractive forces between molecular chains of the plastic and prevent, or minimize, physical bonding of the molecules. Because of the large number of plasticizers available, and the fact that each imparts specific structural values to the base plastic, it is common to use more than a single plasticizer for a specific application.

In general, the more plasticizer added to a resin, the greater the flexibility. It is not uncommon for some grades of flexible PVC to contain as much as 60% plasticizer. The most common plasticizers are phthalates, epoxies, adipate diesters, sebacates, and polyesters.

Heat Stabilizers

Heat stabilizers consist of such materials as lead, organotin, cadmium, tin salts, and others. They are used for preventing degradation of the plastic material, either while being processed or in end-use environments. Because PVC is extremely heat-sensitive, especially when exposed to normal injection molding process temperatures, it is the largest consumer of heat stabilizers of all plastics. However, there are other plastics, such as some grades of ABS, chlorinated polyethylene, and certain blends of various resins with PVC, that do require the use of heat stabilizers, mainly for processibility. Heat stabilizers are available in both solid and liquid form and are usually blended in the initial compounding phase of resin manufacture.

Antioxidants

Plastics tend to deteriorate when exposed to heat, light, or various chemicals. Such deterioration results in brittleness, unstable melt flow properties, loss of tensile properties, and discoloration. *Antioxidants* are used to prevent or minimize this deterioration through absorption or screening of ultraviolet light, deactivation of metal ions, and decomposition of hydroperoxides to nonradical products. Antioxidants can be combined to result in multiple approaches to preventing deterioration.

Some of the more common organic chemicals used as antioxidants are phenols, arylamines, phosphites, and various sulfur compounds. These antioxidants work by greatly reducing the normal tendency of oxygen to combine chemically with hydrocarbons, the basic building blocks of plastic materials. Their use can increase the end-use life of a plastic product by up to 10 times.

Flame Retardants

While *thermoset* plastic materials are inherently flame retardant, *thermoplastic* plastic materials must be compounded with additives to provide flame retardancy. The most common products used for flame retardancy are both organic and inorganic and include such items as antimony, boron, halogens, bromine, nitrogen, and phosphates.

Flame retardants work on four basic principles: they insulate, create an endothermic cooling reaction, coat the base plastic (thereby excluding oxygen), and react with other materials to combat combustion. Most products must meet some type of flammability rating, usually specified by Underwriters Laboratories (UL) in the U.S.

Ultraviolet (UV) Light Absorbers

Sunlight, the major source of ultraviolet light, eventually degrades virtually every plastic material. This degradation is manifested by discoloration and loss of physical properties of the plastic product. *UV light absorbers* are used to stabilize color and lengthen the life of a product by minimizing these degradation effects.

Black (in any form, whether paint, dye, or carbon black) is the most effective UV absorber, but its use is limited to products that can be black in color. Common UV absorbers include benzophenones (especially for clear materials), alkylated hindered phenol, and benzotriazoles. Only a small amount (approximately 1% by weight) of material is needed, but UV absorption capability is depleted as the plastic ages.

Antistatic Agents

Antistatic agents (also called destaticizers) are used to reduce the buildup of electrostatic charges on the surface of plastic products or materials by increasing surface conductivity. The most common antistatic agents for plastics include amines, quaternary ammonium compounds, phosphate esters, and polyethylene glycol esters. They are usually classified on the basis of level of concentration required and their effectiveness at low humidity.

Blowing Agents

A *blowing agent* (also called foaming agent) is used to produce a cellular structure in a plastic mass, thereby reducing the density and weight of the product. Although a wide variety and number of products and methods are used for foaming, chemical blowing agents are the most common and range from simple salts (such as ammonium or sodium bicarbonate) to complex nitrogen-releasing materials. Nitrogen-releasing compounds dominate the field of chemical blowing agents and consist of such products as azodicarbonamide, the N-nitroso compounds, and sulfonyl hydrazides. Blowing agents can be compounded to the base plastic resin or blended as an additive at the molding machine.

Lubricants

Lubricants are used to enhance resin processing characteristics and to improve the surface appearance of molded products. They generally fall into five major categories: metallic stearates, fatty acid amides and esters, fatty acids, hydrocarbon waxes, and low-molecular-weight polyethylene.

Too much lubrication will result in excessive screw slippage and lower outputs, while too little lubrication can result in thermal degradation and higher melt viscosities. An improper balance of lubricant used with a stabilizer may result in plateout or migration of pigment from the melt.

While the preceding section is not intended to be a comprehensive list of fillers, it points out the variety of products available for compounding plastic materials with virtually unlimited numbers of properties and processing characteristics. Remember that, with few exceptions, the addition of a filler will increase the cost and reduce the processibility of a plastic material.

SUMMARY

Over the years, additives for plastics have been developed to improve one or more of the properties of the pure polymer. They fall into one of two categories: reinforcements or fillers.

Reinforcements are additives that have been used to enhance the physical properties of a polymer, while fillers are used to enhance properties other than the physical strength of a polymer.

Fiberglass is the most common reinforcement for plastics in use today. It is used in quantities ranging from 5 to 80% by weight of the total amount of plastic material.

Reinforcement additives usually add weight to the finished polymer but reduce the IZOD impact results due to the test being performed at right angles to the reinforcement orientation.

Fillers are used to enhance other than physical properties and are used at a rate ranging from 10 to 50% by weight.

Generally speaking, two types of colorants are used for plastics: dyes and pigments. The difference between them is their degree of solubility, with pigments being insoluble.

Flexibility in a plastic is achieved by using plasticizers during initial compounding. There are hundreds of plasticizers available, and PVC consumes 80% of those used today.

Most thermoplastic materials are inherently flammable. Flame retardants are added to the polymer to make them flame-retardant to varying degrees. Flame retardants work on four basic principles: they insulate, create an endothermic cooling reaction, coat the plastic (thereby excluding oxygen), and react with other materials to combat combustion.

Blowing agents are used to produce a cellular structure in a plastic mass, thereby reducing the density and weight of the product.

QUESTIONS

1. What two categories of materials are classified under the term “additives”?
2. Why are reinforcements used?
3. Why are fillers used?
4. What is the most popular reinforcement additive used today?
5. How will the addition of glass reinforcement affect the weight of the plastic product?
6. When testing a glass-reinforced material for IZOD impact, will the result be higher or lower than for an unreinforced grade?
7. To what does the term “aramid” refer?
8. What do carbon fiber reinforcements and Thomas Edison have in common?
9. What materials are typically used for producing ceramic fibers?
10. Name three fillers commonly used to increase thermal resistance of a polymer.
11. In general, what are the two types of colorants used for plastics?
12. In general, when plasticizer content is increased, what happens to the plastic?

Comparing Amorphous and Crystalline Materials

4

IDENTIFYING THE DIFFERENCES

Thermoplastic materials are usually categorized as being either amorphous or crystalline. Actually, they also can be in between (semicrystalline), and quite a few plastics can be both. There is also a form called liquid crystalline. But for our purposes, we can consider most plastics to be either amorphous *or* crystalline. As we saw in Chapter 2, there are major differences in physical, mechanical, electrical, thermal, and visual properties between amorphous and crystalline materials. In addition, there are significant differences in processibility. Because the molecules in amorphous materials begin to move as soon as they are exposed to a source of heat, they process over a wider range of temperature settings than crystalline materials which must be brought up to an exact, specific temperature before the molecules begin to move. Of course, we need this movement of the molecules because that is what makes the plastic material soft enough to inject into a mold.

Molecular Structure

The major difference between amorphous and crystalline materials is in their molecular structure. While it is not the intent of this book to provide extensive chemical engineering data, it is necessary to discuss some of the basic chemistry of plastics to better understand when one plastic should be used over another.

Crystalline Polymers

The molecular chain structure of crystalline plastics (Figure 4-1) consists mainly of carbon-to-carbon single bonds. This results in the chains being very flexible, which in turn allows them to fold up on themselves and pack with other chains, forming very ordered areas called crystals.

Forces between the atoms hold the crystals stable and create a level of crystallinity ranging from 0 to 100%. This “degree” of crystallinity is determined by the time and temperature parameters used during the molding process. A crystalline plastic must be heated above its melting point to break down the forces holding the molecules together. At that point it melts and is ready for molding. Then the plastic must be cooled slowly to allow the crystals to re-form. This is

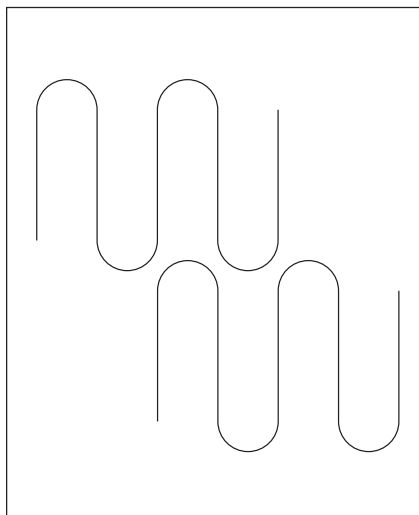


Figure 4-1. The ordered symmetry of crystalline molecular structure.

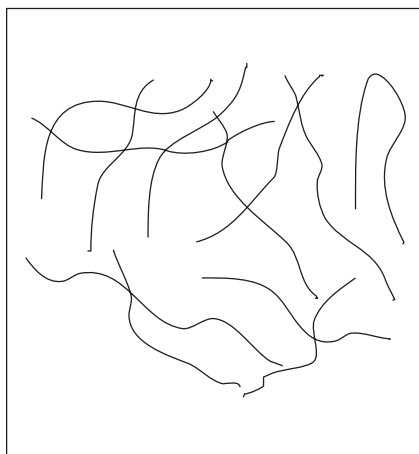


Figure 4-2. Amorphous molecular structure is unstable and random.

possible because the molecules are mobile for a long period of time, seeking each other out and linking themselves together. The faster the plastic is cooled, the quicker the molecules slow down, and the lesser the linking activity between molecules; thus the lower the level (degree) of crystallinity. The level of crystallinity has great influence in determining the properties of the molded product.

Amorphous Polymers

Amorphous plastics tend to be stiffer than crystalline and do not fold up tight enough to form crystals. They are also bulkier and usually have large groups of atoms bonded to the main chain. Secondary forces are not sufficient enough to maintain stability. The molecular structure remains random at all temperatures, as shown in Figure 4-2.

When the amorphous plastic is heated, its molecular structure does not break down or change as in crystalline plastic. Rather than truly melt, the amorphous material simply begins to soften. It will become softer and softer as more heat is applied, until it is soft enough to mold.

Crystalline/Amorphous Polymers

While these polymers are not used much, they do exist and we show their molecular structure. Notice in Figure 4-3 that there are both amorphous and crystalline areas.

The crystalline areas are connected to the amorphous areas by atomic forces, but the combined areas are only loosely linked to each other. The resultant polymer has some of the advantages and disadvantages of both the crystalline and amorphous polymers.

Process Temperature Requirements (Injection Unit)

Because of the differences in their basic molecular structure, crystalline materials respond differently to temperature than amorphous materials. This difference is due to the mechanisms required to break down the natural affinity of polymer molecules to each other.

In the crystalline materials the molecular chains are very structured. This was shown in Figure 4-1. When heat is applied (as is done in the injection barrel of a molding machine), the plastic begins to absorb the heat and the molecules begin to react to that heat absorption by slowly moving about. Their movement is curtailed, however, by the rigid structure and forces holding the chains together. As more heat is applied, more movement is attempted, until, finally, the chains do break apart, and the plastic becomes fluid. This is represented in Figure 4-4.

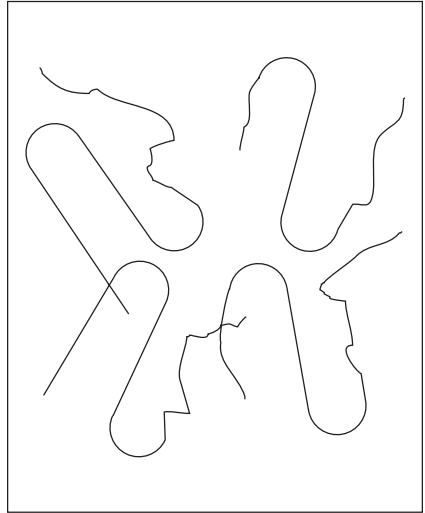


Figure 4-3. Crystalline/amorphous molecular structure.

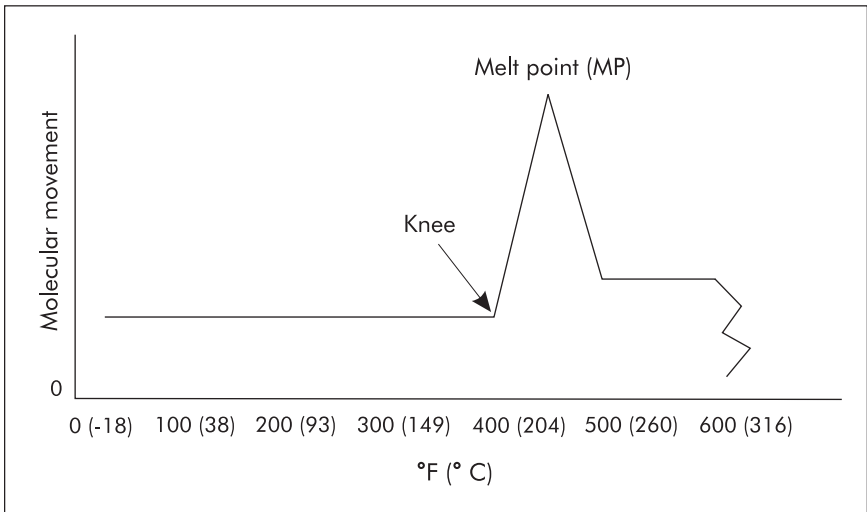


Figure 4-4. Differential Scanning Calorimeter graph of crystalline material.

The graph in Figure 4-4 was generated by a test machine called a Differential Scanning Calorimeter (DSC). The DSC measures the molecular movement of a plastic material (as a function of the number of calories expended or absorbed) compared to the amount of energy needed to move the molecules. This graph indicates that as the temperature increases (across the bottom of the graph), molecular movement is stable until the temperature reaches what is called the *melt point* (MP) of the crystalline material. At that point, the molecular chain connections break and the plastic becomes fluid, or melts. The shape of the peak shows that there is some molecular action beginning at around 380° F (193° C) then building quickly to the top of the peak at approximately 420° F (216° C). That is the point (MP) at which the plastic becomes fluid, and the molecular motion stabilizes (at approximately 475° F [246° C]) as the temperature increases until degradation occurs at approximately 600° F (316° C). At this point of degradation, the molecules begin to break down into their main atomic elements, which results in formations of carbon to varying degrees. It may become apparent that the ideal molding temperature for crystalline materials is the melt point. That is the temperature at which the injection barrel should *maintain* the plastic material. The temperature range between the first and last “knee” of the peak is the range within which the plastic should be molded to attain the best properties and the best processibility. To maintain that range, the barrel of the injection machine would actually be set at temperatures approximately 50° F (10° C) higher. This is because of the steady loss of heat as material passes through the barrel.

With amorphous materials, the DSC graph takes on a different appearance, as shown in Figure 4-5.

The figure shows that the molecular movement begins immediately as heat is applied. This movement simply continues to increase as more heat is applied, until degradation occurs. This is because of the loose molecular structure of amorphous materials. These materials do not exhibit the tight and orderly structure of the crystalline variety, so the molecular chains are free to move with little constraint. But there still is an ideal temperature at which the amorphous materials should be molded. That temperature can be determined by analyzing the DSC curve closer. In Figure 4-6 you can see that by magnifying a specific area of the DSC line we find what looks like an “S” formation. That area is where the amorphous material changes physically from a “putty” consistency to a very fluid consistency.

This is called the *glass transition*, or T_g point. By drawing a vertical line through both curves of the S and extending it to the temperature axis, we can determine the actual, ideal glass transition temperature. Again, that is the *ideal molding temperature*. Because amorphous materials have such a wide range of softening, they can be molded over a very wide range of temperatures. The

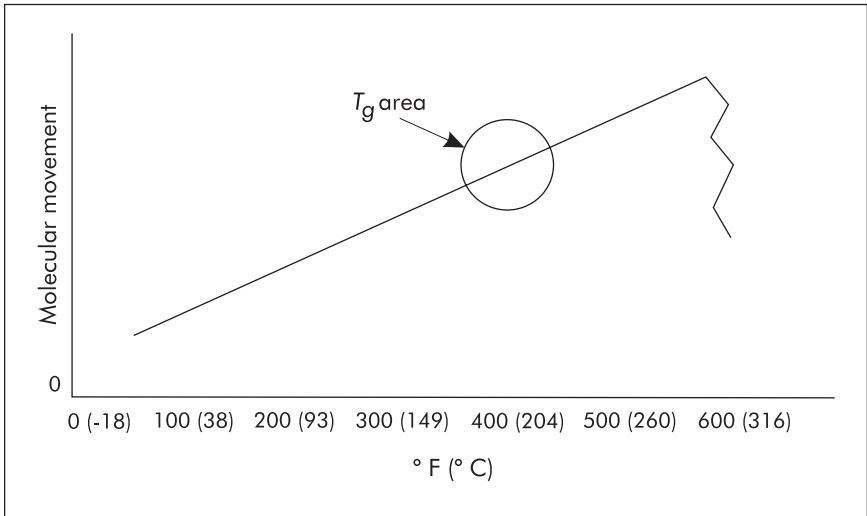


Figure 4-5. DSC Graph of amorphous material, highlighting the glass transition area.

closer that temperature is to the glass transition temperature, the better the properties of the molded product.

Process Temperature Requirements (Mold)

The temperature of the mold used for injection molding is as important as the temperature of the injected plastic. The mold temperature is controlled (as discussed in Chapter 1) by circulating water (or oil if the temperature requirement is over 200° F [93° C]) through channels that are machined into the mold itself. If the temperature is too low, the plastic solidifies too soon and will not fill the mold. If the temperature is too high, the mold must stay closed longer while waiting for the plastic to solidify, and the cycle time becomes too long. The temperature must be maintained somewhere between these two conditions. The actual mold temperature requirement depends on many variables, one of which is whether the plastic is crystalline or amorphous.

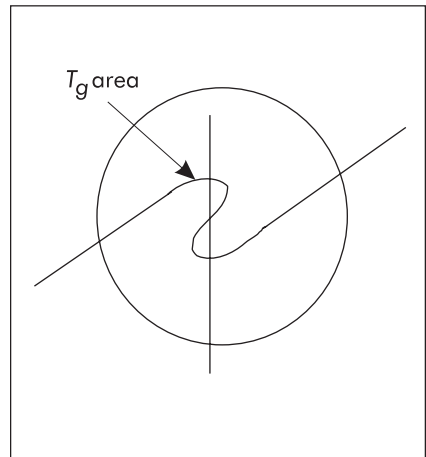


Figure 4-6. Determining the glass transition point on the DSC graph.

Crystalline

Remember that crystalline molecular chains break down when exposed to temperatures required for injection molding. The molecules stay this way while being injected into the mold but immediately begin to re-form their crystalline order as the temperature drops after the plastic enters the relatively cold mold. This re-forming of structure is slow, however, compared to the rate at which the plastic may solidify if the mold is too cool, and the molecules may not completely re-form. The warmer the mold, the longer the time allowed for the re-formation to take place. Crystalline materials acquire their property values from the formation of crystalline molecular structure. The less crystalline structure there is, the lower the value of those properties.

Table I-1 in Chapter 1 shows the recommended mold temperatures for common crystalline materials. For those not shown, the material supplier can provide the information.

It is not practical to assume that a plastic product must remain in the properly temperature-controlled mold long enough to re-form to 100% crystallinity. For this to happen would require extremely long cycle times which would not be economically feasible. Tests have shown that property values are within acceptable limits when crystallinity is at or above 85%. This has become an accepted standard and one that can be tested for using DSC equipment. The test consists of running a known sample that was intentionally produced to achieve the 100% crystallinity value and comparing it with a sample from a production run. An example is shown in Figure 4-7.

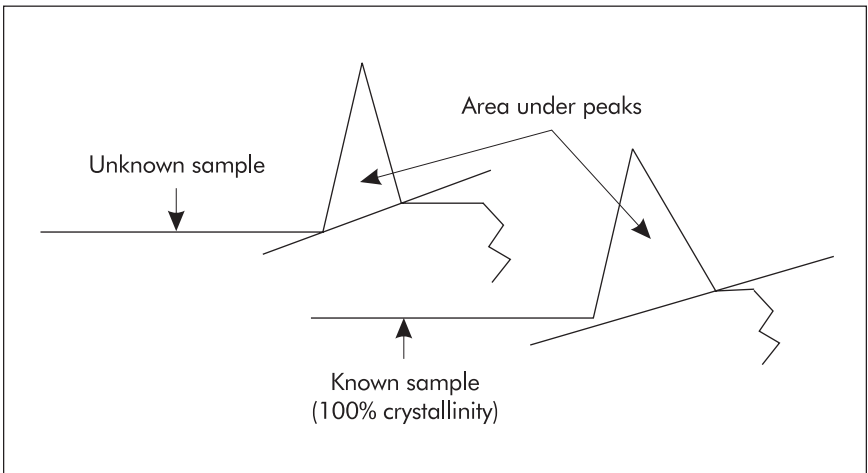


Figure 4-7. Determining the percentage of crystallinity using DSC.

The area under the peaks of both samples is compared and the resultant ratio indicates the percentage of crystallinity attained by the production process. If the ratio shows less than 85% crystallinity, the production process should be adjusted accordingly. Usually that requires a higher mold temperature, a longer cycle time, or both. Of course, it may be determined that less than 85% crystallinity is acceptable for a specific product and a given environment. That must be determined during the product design cycle and specified so the producer (molder) of the product is aware that the processing window is larger than normal. This should result in lower cost to the customer.

Amorphous

Amorphous materials are much more forgiving than crystalline materials in terms of mold temperature requirements. This is because the original molecular structure doesn't really break down at exposure to temperature, as the crystalline molecules do. Instead, amorphous molecules simply begin to move when exposed to heat, and move more as more heat is applied. Upon cooling, the molecules slow down until solidified. Although the effect is less than that found with crystalline materials, the rate at which amorphous materials cool down does affect their properties, too.

General Differences Based on Crystallization

After being heated to the proper molding temperature, both amorphous and crystalline materials tend to react the same during the injection phase of the molding process. From that point forward, however, they react differently as they cool and solidify. Some of these differences are:

- The ordered structure of the crystalline materials allows the molecules to come in closer contact with one another. The result is that the crystalline materials take up less volume than amorphous materials. Therefore, *crystalline materials exhibit more shrinkage than amorphous materials.*
- The percentage of crystallinity achieved determines the degree of shrinkage. The variation in molding conditions can create fluctuations in the percentage of crystallinity achieved. Therefore, *it is much more difficult to hold tolerances with crystalline materials than with amorphous.*
- The intermolecular forces holding together the highly structured crystalline molecules are much greater than those holding together the random-structured amorphous molecules. It takes much more energy to break the crystalline bonds than the amorphous bonds. It also requires more energy to re-form the crystalline bonds during the cooling process. Therefore, *it is more expensive to process the crystalline materials.*
- Because crystallinity exhibits a very compact molecular structure, the density of the plastic increases as the percentage of crystallinity increases,

which inhibits molecular segment rotation. Flexibility is determined by the ability of the molecular segments to rotate. Therefore, *crystalline materials tend to be stiffer than amorphous materials because of the inability of the crystalline molecular segments to rotate.*

- The compact structure of the crystalline molecules results in higher tensile strength than with amorphous materials. *The greater the crystalline percentage, the greater the tensile strength.*
- Amorphous materials have a greater impact strength because they do not exhibit the fault lines that are so common with crystalline molecular structures. These fault lines increase with crystallinity percentage. Therefore, *the greater the crystallinity percentage, the lower the impact strength.*
- *The heat resistance properties of crystalline materials are higher than amorphous materials* because the crystalline materials must absorb much more heat before the molecular structure begins to break down.
- Increased crystallinity percentage brings the molecules closer together. This increases resistance to permeability of gases and vapors. Therefore, *amorphous materials do not possess the high degree of permeability resistance common with the crystalline materials.*
- *Amorphous materials have a greater resistance to stress cracking than crystalline materials* owing to the absence of the same fault lines that lower the impact strength of crystalline materials.
- Because *crystalline materials* are made up of many different areas exhibiting varying densities (therefore shrinkage rates), they *tend to warp more than amorphous materials* which have more consistent density areas throughout the polymer. The varying densities found in the crystalline materials set up internal stresses that result in warpage, twisting, and bowing.
- Crystalline materials remain solid until their melt point is reached, at which time they melt completely and are ready to flow. Amorphous materials never really melt, but get continuously softer as more heat is applied. Because of this phenomenon, the *amorphous materials do not flow as easily as the crystalline materials.*

Liquid Crystalline Materials

Another group of plastics becoming more and more popular are the liquid crystal materials. These materials have the high melt temperature requirements of crystalline plastics but soften gradually and continuously like amorphous polymers. They have the lowest viscosity, warpage, and shrinkage of all thermoplastics. Table IV-1 shows the comparison of some common properties of crystalline, amorphous, and liquid crystalline materials.

Table IV-1. Comparison of Crystalline, Amorphous, and Liquid Crystalline Materials

Property	Crystalline	Amorphous	Liquid Crystalline
Specific gravity	Higher	Lower	Higher
Tensile strength	Higher	Lower	Higher
Tensile modulus	Higher	Lower	Higher
Elongation	Lower	Higher	Lower
Creep resistance	Higher	Lower	Higher
Maximum use temperature	Higher	Lower	Higher
Shrinkage	Higher	Lower	Lower
Warpage	Higher	Lower	Lower
Flowability	Higher	Lower	Higher
Chemical resistance	Higher	Lower	Higher

SUMMARY

Although there is a form called semicrystalline, most plastics can be considered either amorphous or crystalline in molecular chain structure.

There are major differences in physical, mechanical, electrical, thermal, and visual properties between amorphous and crystalline materials as well as processibility properties.

Physical properties of a molded crystalline material are greatly reduced if proper recrystallization is not achieved. This happens when a part is cooled too quickly, such as in a chilled mold.

DSC can be used to determine many properties of plastic materials, including the melt point of a crystalline material and the glass transition point of an amorphous material.

Amorphous materials have a greater resistance to stress cracking and warpage than crystalline materials.

Crystalline materials have a greater resistance to heat and higher tensile strength than amorphous materials.

Liquid crystalline materials have the lowest viscosity, warpage, and shrinkage of all thermoplastics.

QUESTIONS

1. Into which two categories are most thermoplastic materials placed?
2. What is the primary cause of the major differences between amorphous and crystalline materials?

3. What single value most determines the physical properties of a molded crystalline product?
4. Instead of literally “melting” what does an amorphous material actually do?
5. What piece of analysis equipment can be used to determine the plastic melt point and glass transition?
6. How is a typical mold temperature controlled?
7. What degree of crystallinity is generally accepted as “standard”?
8. In general, why do crystalline materials shrink more than amorphous materials?
9. Why is it usually more expensive to process crystalline materials than amorphous materials?
10. How do liquid crystal polymers rate concerning viscosity, warpage, and shrinkage?

Basic Material Properties

PROPERTY DESCRIPTIONS AND TERMINOLOGY

A host of different properties are associated with any material, including plastics. The following paragraphs describe some of the more common properties, some of which are peculiar only to plastics. They are listed in the categories of physical, mechanical, thermal, electrical, and environmental. (Descriptions and identification of tests used to determine these properties are discussed at length in Chapter 9.)

Physical Properties

Density

Density is defined as the measure of mass per unit volume of any material, expressed as pounds per cubic inch (lb/in.³), or grams per cubic centimeter (g/cm³). This property is used to calculate the relationship between weight and volume of plastic in a specific part design. The density value tells what each cubic inch (or cubic centimeter) of the part weighs.

Specific Gravity

Specific gravity is a ratio that compares the density of any material divided by the density of water, both measured at 73° F (23° C). The formula is:

$$\text{specific gravity} = \text{density of material} \div \text{density of water}$$

It, too, (like density) is used for determining part weight and cost.

Shrinkage Rate

Although it is a misnomer (*rate* implies a time value, which is not present), this term is used to define the expected reduction of the plastic part dimension as the part cools (and solidifies) in the mold, compared to the original mold dimension, Figure 5-1.

Shrinkage rate is usually given as inch per inch (in./in.), or centimeter per centimeter (cm/cm), and the range is between zero for low shrinkage materials, and approximately 0.050 in./in. (0.127 cm/cm) for very high shrinkage materials. Actual shrinkage values depend on many variables such as temperature, pressure, injection time, and wall thickness of the part. In general, amorphous materials have lower shrinkage than crystalline materials, and glass-reinforced

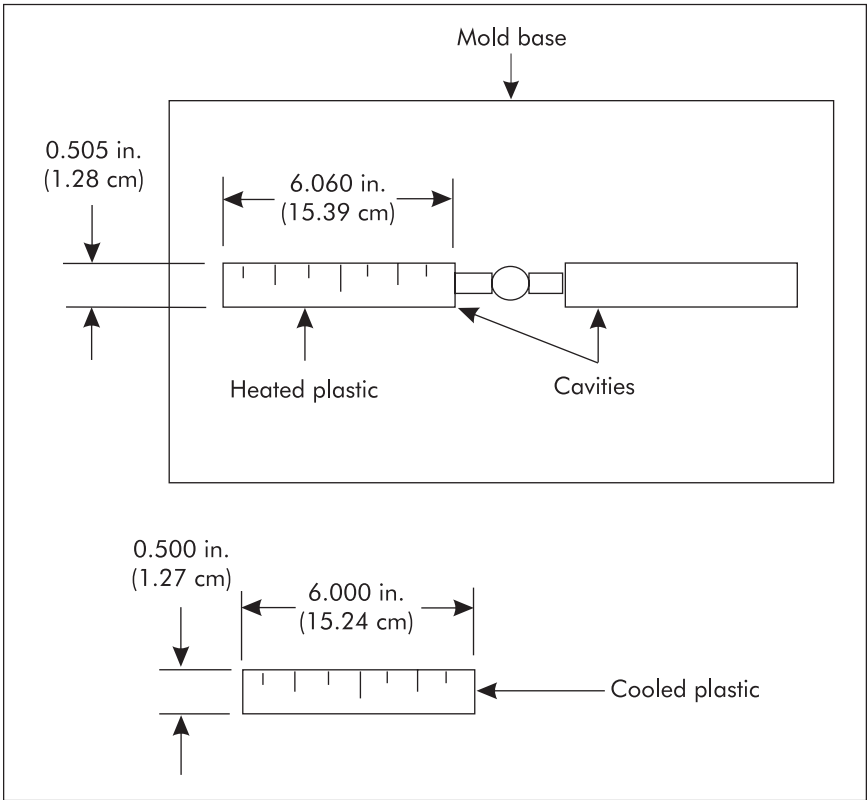


Figure 5-1. Shrinkage rate (mold shrinkage).

or filled materials have less shrinkage than unfilled, or *neat* resins. Sometimes the shrinkage rate is given as a percentage, with each % equal to 0.010 in. (0.025 cm).

Water Absorption

This property is expressed as the percentage increase in weight of a material due to absorption of water. A test determines this percentage by comparing the weight of a fully dried plastic specimen to that same specimen after being immersed in water for varying time periods. This property determines the effects on mechanical and electrical properties of a specific plastic. Also, plastics with low absorption rates tend to be more dimensionally stable.

Transparency (Opacity)

Properties of transparency and opacity determine light transmission abilities of a given plastic and are usually measured as *haze* and *luminous transmittance*.

Haze is defined as the percentage of light transmitted through a test specimen that is scattered more than 2.5 degrees from the incident light. Luminous transmittance is defined as the ratio of transmitted light to incident light.

Modulus (Toughness)

Modulus is a measure of flexibility and refers to the material's ability to absorb mechanical energy without fracturing. Toughness is usually measured by the area under the stress/strain curve of a plastic test specimen as shown in Figure 5-2.

In general, high-impact, unfilled plastics have excellent toughness values.

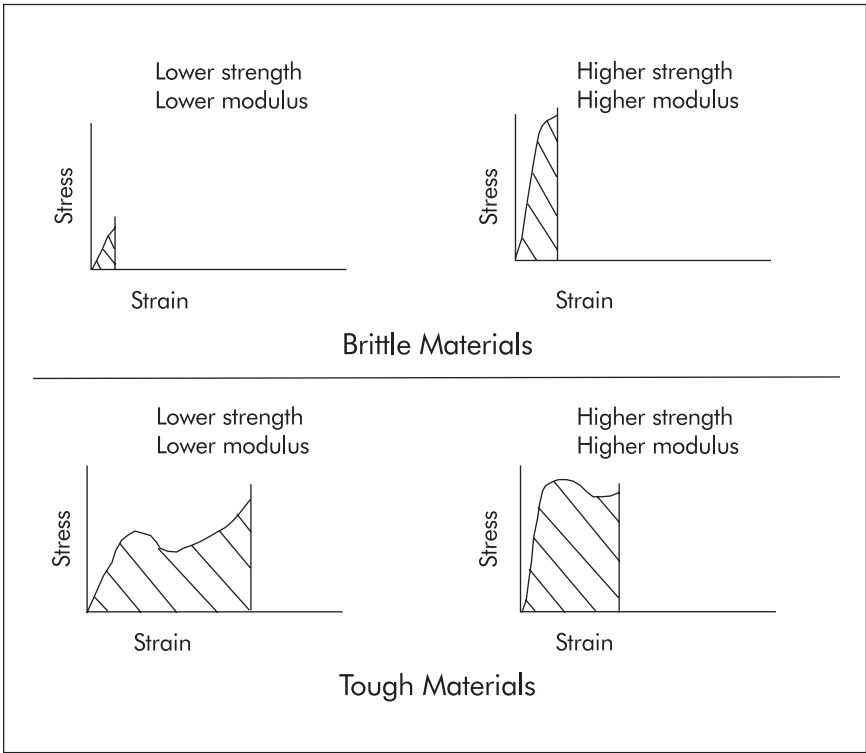


Figure 5-2. Stress/strain curves.

Elasticity

Elasticity is the ability of a material to return to its original size and shape after being deformed, as shown in Figure 5-3. Most plastics (other than elastomers) have very limited elasticity.

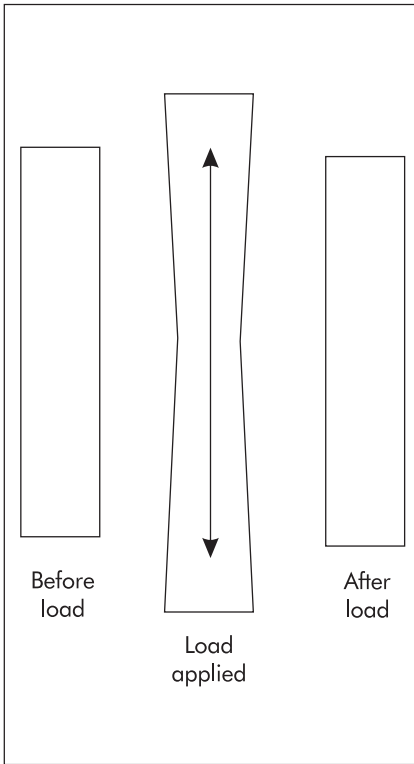


Figure 5-3. Material exhibiting elasticity.

Brittleness

Brittleness is actually a lack of toughness, and it can be exhibited through lower impact strength and higher stiffness properties. Usually, the glass-reinforced and mineral-filled plastics are especially brittle, but brittleness can also be a function of too much moisture or improper heat settings during processing. Hygroscopic materials (those that absorb moisture from the atmosphere) will appear brittle after molding until they achieve moisture equilibrium with the surrounding environment.

Plasticity

Plasticity is the inverse of elasticity in that it is a value of the ability of the material to stay in the size and shape to which it is deformed. Plastic materials exhibit plasticity when they are stressed beyond their yield point. Some plastics can be easily cold-formed owing to this property. Figure 5-4 shows the result of a material undergoing plasticity.

Ductility

To be ductile, a material must exhibit properties allowing it to be stretched, pulled, or rolled into shape without destroying its structural integrity.

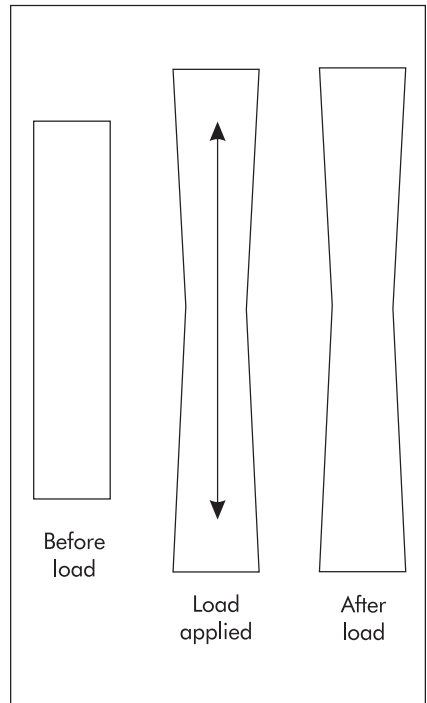


Figure 5-4. Material exhibiting plasticity.

Notch Sensitivity

Notch sensitivity is a measure of the ease with which a crack propagates through a plastic part from a pre-existing notch, crack, or sharp corner.

Lubricity

Lubricity is a function of the load-bearing characteristics of a material under relative motion. It refers to the ability of a plastic material to slide against itself or other plastic materials. Plastics with good lubricity have less tendency to wear away through friction.

Isotropy and Anisotropy

Isotropy refers to the fact that physical properties of a plastic material are the same regardless of the direction in which they are measured, shown graphically in Figure 5-5. Anisotropy refers to the fact that physical properties of a plastic material will vary depending on the amount of symmetry that exists and the direction in which the property was measured. This is shown in Figure 5-6.

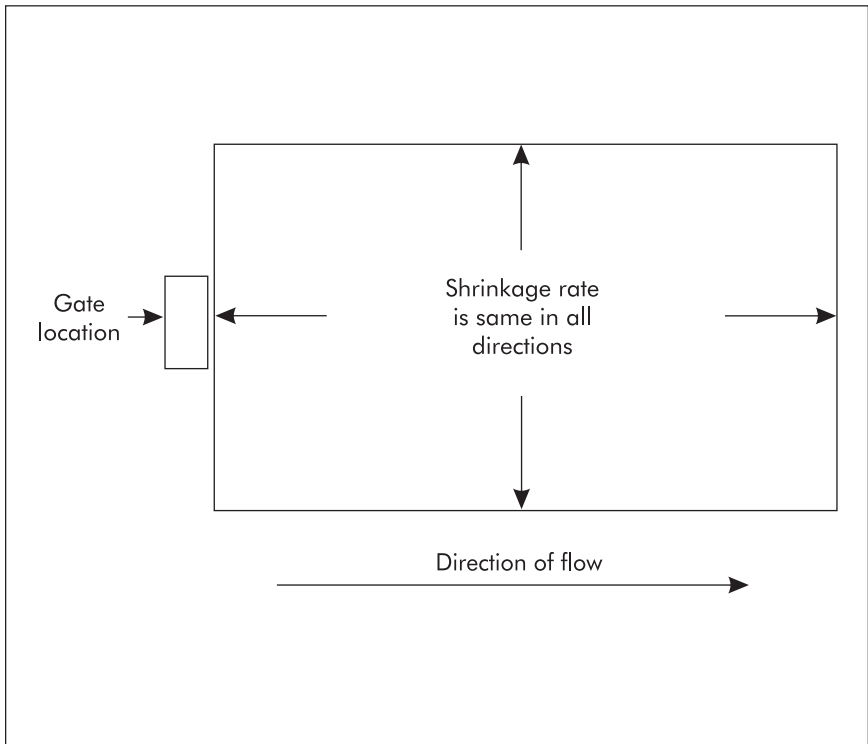


Figure 5-5. Isotropic measurement.

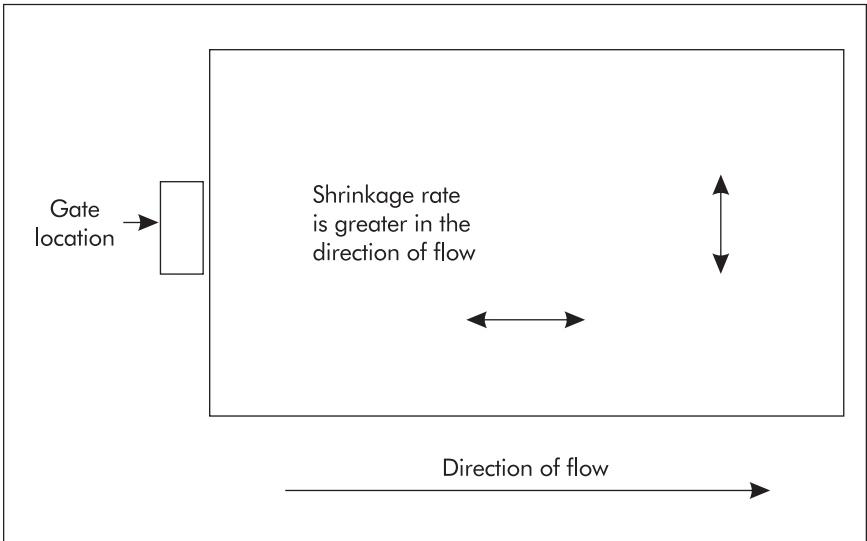


Figure 5-6. Anisotropic measurement.

Mechanical Properties

It must be noted at the onset that there is a danger of placing undue emphasis on individual mechanical properties of plastic materials. Usually, there is no single property that is distinctive in determining the reaction of a plastic to a specific set of conditions. Published values of mechanical properties are established through use of individual, very specific tests, performed under critical laboratory conditions. Yet, the finished plastic product is seldom exposed to such rigorous individual situations. The mechanical stresses presented to a plastic part in everyday use are actually combinations of stresses, all working with or against each other, therefore influencing each other. These facts must be taken into consideration when attempting to select or understand a plastic material based on mechanical properties. That being understood, the published mechanical properties can certainly be used as guidelines and indicators of a plastic's ability to withstand certain exposures to use. The following discussion is a combination of definitions and concepts of mechanical properties of plastic materials.

Stress and Strain

There are essentially two types of stress: direct and shear. *Direct stress* is reflected in forces that act perpendicular to a surface. *Shear stress*, on the other hand, is reflected in forces that act parallel to that surface. Usually, when stress is present, strain is present also. Strain is a ratio measure between a change in dimension of a stressed plastic versus the original dimension of the plastic before

being stressed. Strain measurements are usually represented as a percentage. Let's take a closer look at stress and strain.

Normal stress. Normal stress is defined as the ratio of applied force to the original cross-sectional area. It is expressed as pounds per square inch (lb/in.²) or psi. It can also be expressed as Newtons per square meter (N/m²), or Pascals (Pa). The basic formula is:

$$\text{stress} = \text{load} \div \text{area}$$

Figure 5-7 is a schematic of normal stress applied to a specimen during a standard tensile test.

The figure shows the load applied in tension mode (sample is pulled apart). If the load application is reversed, the test is performed in compression mode (sample is pushed together).

Normal strain. Normal strain is a measure of the deformation (change in length) of a material subjected to a direct load. The formula is:

$$\text{strain} = \text{change in length} \div \text{original length}$$

with the result expressed as a percentage. To understand the relationship between stress and strain, we perform a *tensile test*. Figure 5-8 shows the operation of this test.

The results of the tensile test are displayed on a curve called the stress/strain curve. A typical stress/strain curve, plotting the results of a test performed in tension mode, is shown in Figure 5-9.

Modulus of Elasticity

Because stress is proportional to load, and strain is proportional to deformation, it is apparent that stress should be proportional to strain. Hooke's Law states that degree of proportionality as:

$$\text{stress} \div \text{strain} = \text{constant}$$

The constant in Hooke's Law is called the modulus of elasticity (usually written as E), and this is sometimes referred to as Young's modulus, or tensile modulus. This modulus is actually a measurement of the "slope" of the original portion of the stress/strain curve. For some materials the result of such a measurement can be misleading, so some suppliers use the "secant" modulus measurement instead. A discussion of all the elements of a typical curve follows (Figure 5-10).

- **Proportional limit.** The proportional limit is the greatest stress at which a material is capable of sustaining a load without deviating from the proportionality of stress to strain. It is the point where the slope begins to change and linearity ends, and is measured in psi (Pa).
- **Yield point.** This is the first point on the stress/strain curve at which an increase in strain occurs without an increase in stress. The slope of the curve is said to be at 0 at this point.

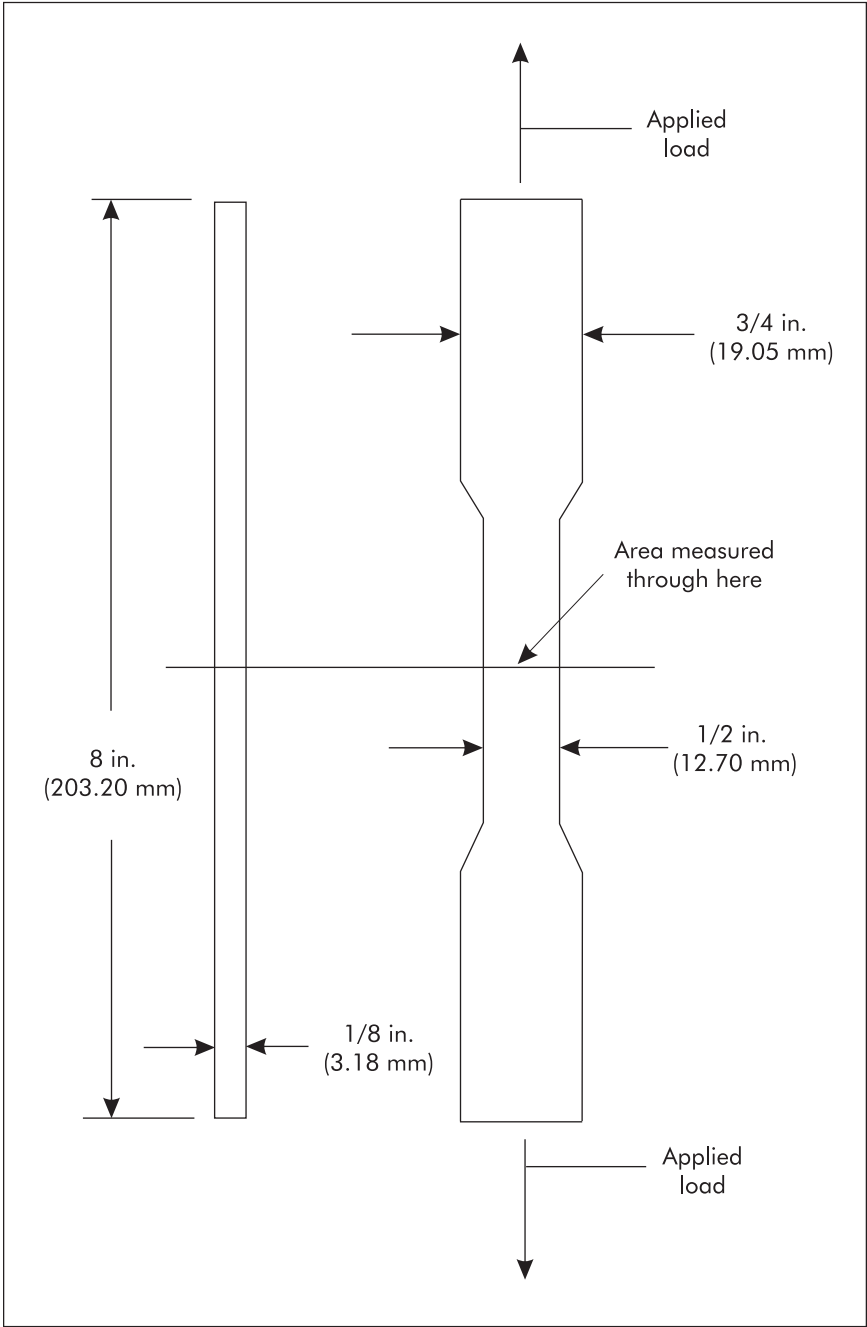


Figure 5-7. Specimen with stress applied.

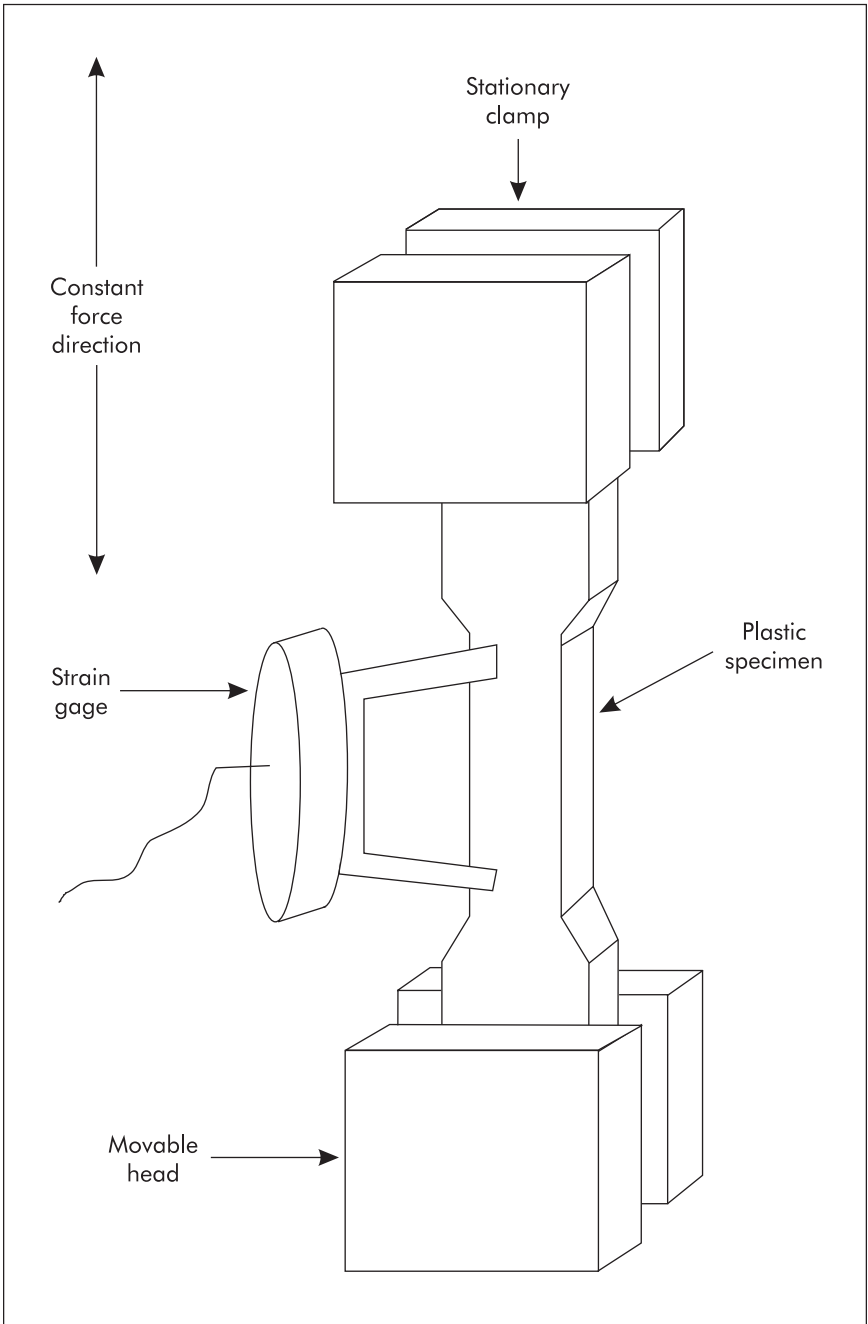


Figure 5-8. Typical stress/strain (tensile) test.

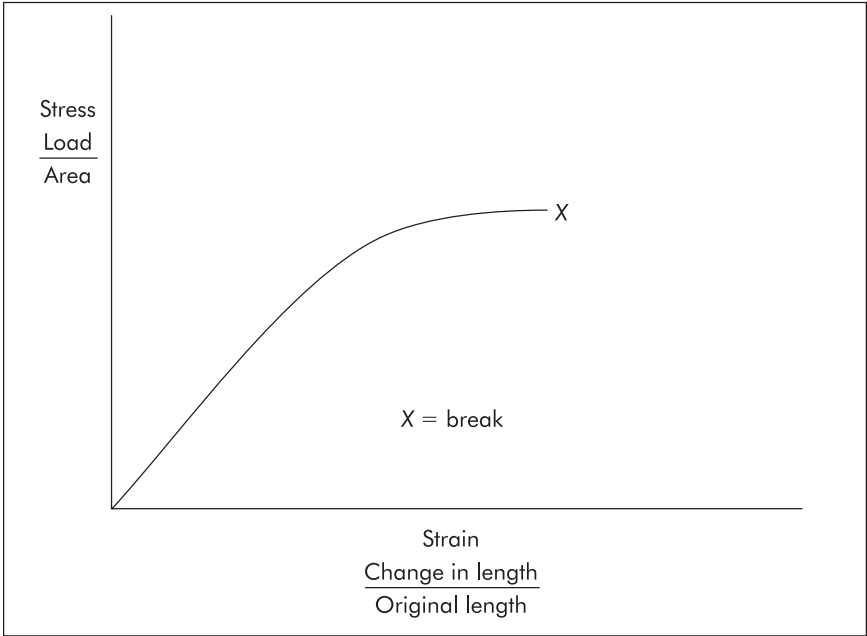


Figure 5-9. Typical stress/strain curve plot.

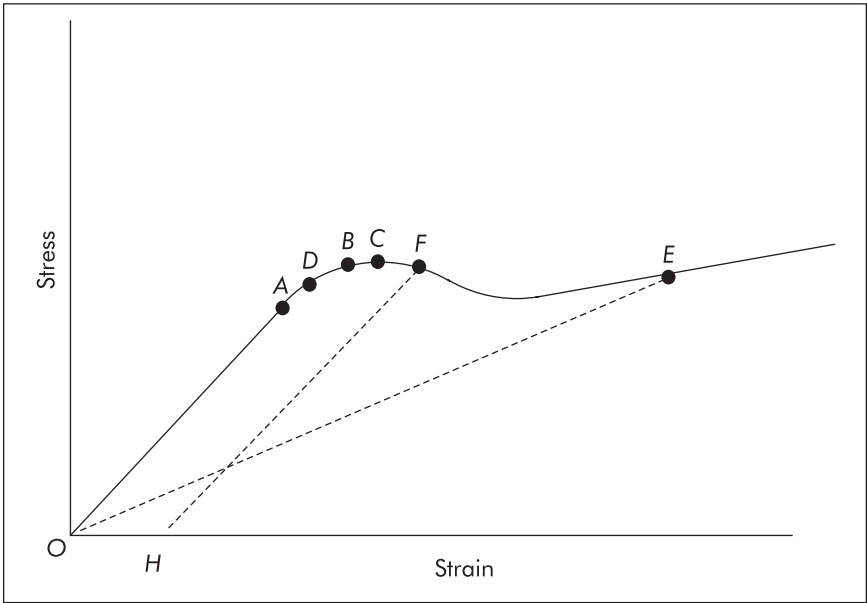


Figure 5-10. Stress/strain curve elements.

- **Ultimate strength.** This is the maximum stress a material withstands when subjected to a load. It is expressed in psi (Pa).
- **Elastic limit.** This point on the stress/strain curve represents the point beyond which the material is permanently deformed if the load is removed.
- **Secant modulus.** The secant modulus is a measurement of slope and represents a ratio of stress to corresponding strain at any point on the stress/strain curve.
- **Yield strength.** For those materials that do exhibit a yield point (*B*), the yield strength can be determined by selecting a stress level beyond the elastic limit. An imaginary line is drawn running parallel to *OA* at a specified offset strain, point *H*. The stress where the line intersects the stress/strain curve at point *F* is referred to as the yield strength at *H* offset. For instance, in Figure 5-10, if point *H* were at 2% strain, point *F* would be identified as the *yield strength at a 2% strain offset*.

Poisson's Ratio

Poisson's ratio is a constant used for determining the stress and deflection properties of plastic structures such as rotating discs or plates. It compares, under load, the lateral compression strain versus the longitudinal stretching strain of a test bar, with the result shown as:

$$\nu = \text{lateral strain} \div \text{longitudinal strain}$$

where:

the quotient of Poisson's ratio is designated by " ν ."

The values of ν usually lie between 0.20 and 0.40 for neat resins, and 0.10 and 0.40 for filled materials.

Shear Stress and Shear Strain

Imagine a solid block of plastic material to actually consist of infinitesimally thin layers. If equal and opposite forces are applied, the layers tend to slide over each other as shown in Figure 5-11.

The formula for shear stress is:

$$\text{shear stress} = \text{shear load} \div \text{area resisting shear}$$

and is always tangential to the area on which it acts.

Where:

shear strain is the angle of deformation (α) and is measured in radians.

Shear Modulus

For materials that behave according to Hooke's Law, shear modulus (modulus of rigidity) is comparable to the modulus of elasticity mentioned. It is a ratio constant determined by dividing shear stress by shear strain.

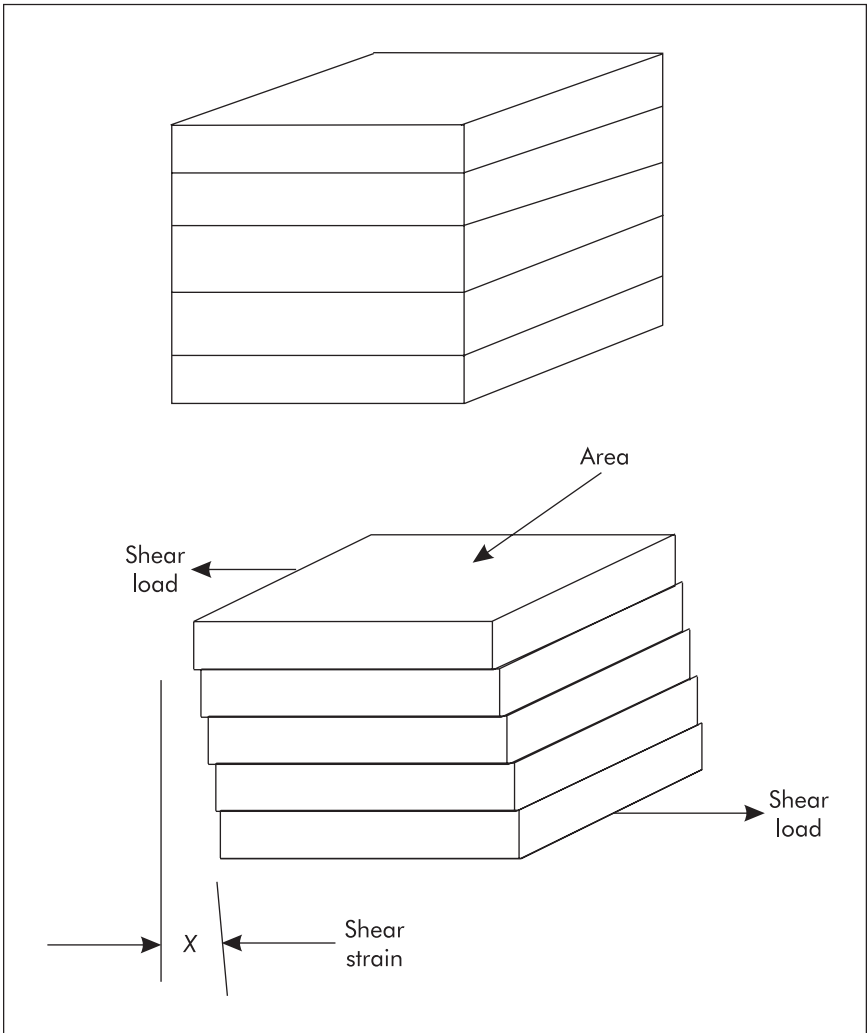


Figure 5-11. Shear stress and shear strain defined.

Direct Shear

While not a true representation of shear as thought of when considering stress/strain curves, marketing personnel often refer to direct shear as an indicator of shear strength of a specific plastic. Direct shear properties should only be considered valid when the end product is designed to be used in an application similar to the test itself, as shown in Figure 5-12.

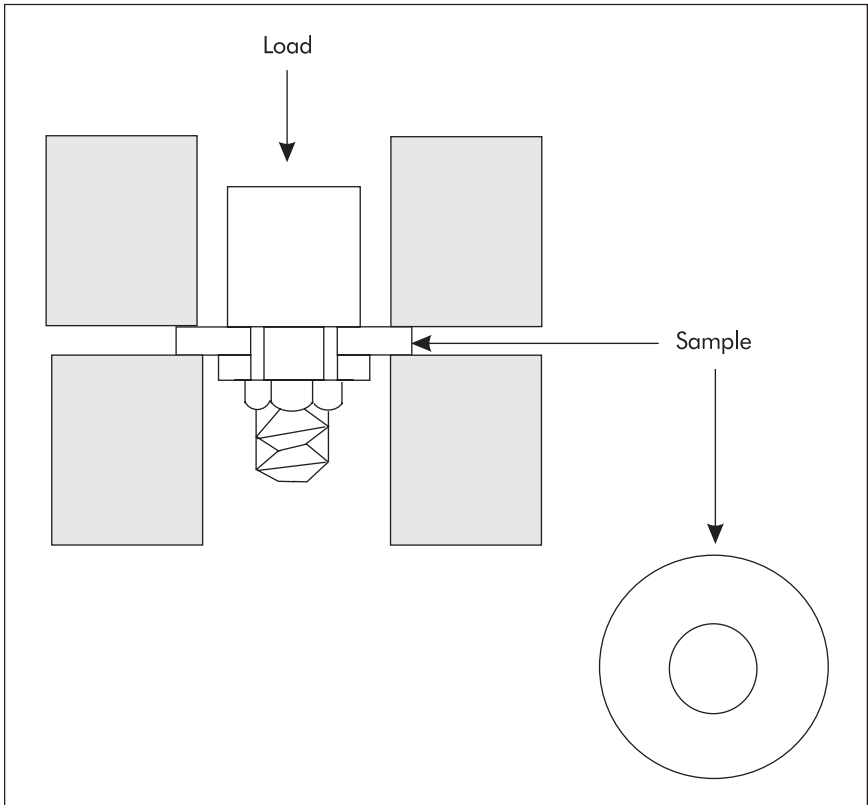


Figure 5-12. Direct shear stress test.

Compressive Strength and Modulus

The elastic modulus of a plastic is usually measured in a tension mode. For most materials the compression and tension curves are almost identical, so tension values are the ones reported. However, at high stress levels compression strain is less than tensile strain. But compression testing does not usually lead to a yield condition, so if it is required, it is reported as the amount of stress required to deform a plastic specimen to a specific strain value. Usually the compressive modulus is not reported because it is equivalent to testing a secant modulus.

Flexural Strength and Modulus

Flexural strength is a property from which we can determine the bending capabilities of a plastic. A specimen is placed across two beams and a force is directed from the opposing side, as shown in Figure 5-13.

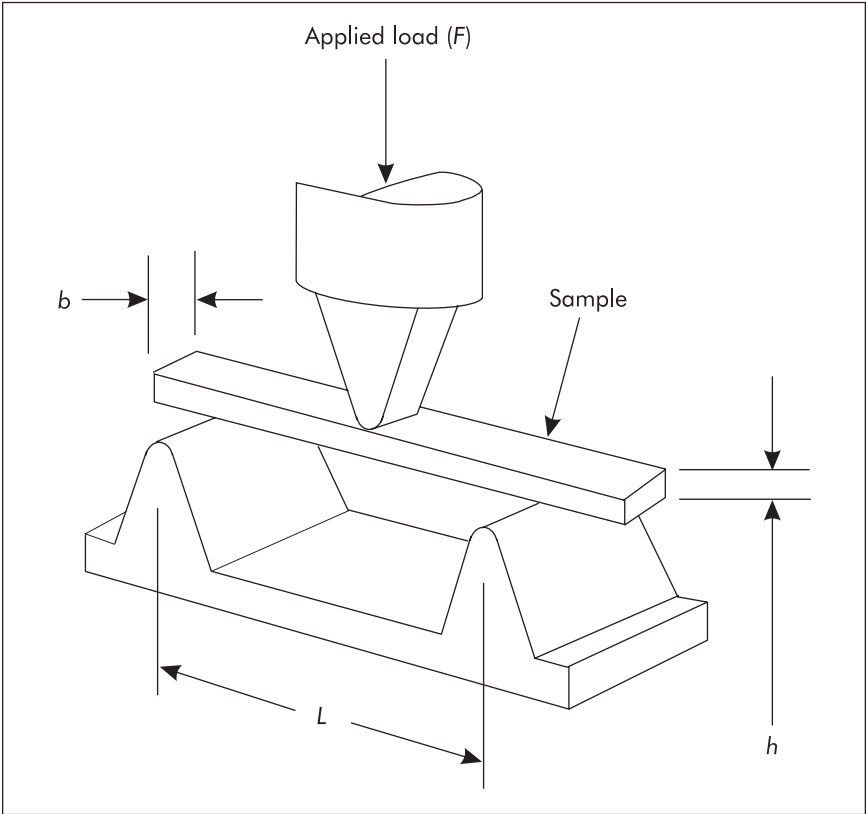


Figure 5-13. Flexural strength fixture.

The formula for flexural strength is:

$$\text{flexural strength} = 3FL \div 2bh^2$$

and the formula for flexural modulus is

$$\text{flexural modulus} = FL^3 \div 4bh^3 Y$$

where:

Y = the amount of deflection at the point of load on the specimen.

Creep

Creep can be defined as increased strain over a period of time in the presence of constant stress occurring from a constant load. A specimen is clamped at one end and a weight is attached to the other. The assembly is suspended and allowed to stabilize over a period of time, ranging from hours to months. The resultant stress/strain-time data is usually presented as creep curves of strain

versus log time. In general, the amorphous plastics have higher creep rates than crystalline plastics, and the addition of glass reinforcement generally improves creep resistance.

Impact Strength

The ability of a plastic part to absorb energy from impact is determined by its basic shape, size, and thickness, as well as the type of plastic used in its manufacture. There are four basic impact tests for determining the *relative* impact strength of a plastic material. (All require a specially molded specimen for the test rather than the part itself.) They are: IZOD impact, Charpy impact, tensile impact, and falling dart impact. We take a closer look at these later in the book. None of these tests provide data that can be used analytically, but they do offer relative impact resistance or notch sensitivity information as an indicator for comparing basic groups of plastic materials.

Fatigue Endurance

Fatigue endurance is the ability of a plastic material to endure repeated stress as the result of cyclic action. A typical example is the snap action latch device, or any component subjected to vibration. A test is performed on a specially-molded specimen, and the number of cycles to failure is recorded for each of various stress levels. The data is presented in a plot of log stress versus log cycles. Heat buildup can contribute to the failure, another of those properties that does not define the end-use limits through use of standard tests, but it can be used as an indicator for material comparison purposes.

Thermal Properties

Anyone involved with selecting a plastic material, such as the design engineer, should be aware of the operating environment to which the final product will be subjected. Thermoplastic materials exhibit varying characteristics dependent on their thermal properties. The following represent some of the thermal properties used for characterizing plastics.

Melting Point

Crystalline materials have a sharp melting point to which the temperature of the plastic must rise before it becomes fluid enough to flow. That temperature is called the melting point. It is defined as the tip of a peak plotted on a curve as the result of performing a DSC test. The test is detailed in Chapter 9. The melting point is referred to as melt point, melt temperature, or T_m .

Glass Transition Temperature

Amorphous and liquid crystalline plastics do not have a sharp melting point but, instead, begin to soften as soon as they are exposed to heat. They continue to

soften as the heat increases until they are soft enough to flow properly. There is a temperature, called the glass transition point, at which the plastic undergoes a significant change in properties. Generally, at temperatures below the glass transition point (referred to as T_g) the plastic is stiff, brittle, and glassy, while above the T_g it is more ductile and has a rubbery response to impact loads.

Melt Index

The melt index (also called flow index, melt flow, and melt flow index) is a value that defines the ability of the plastic material to flow. It is usually expressed in grams per 10 minutes and reflects how much material flows through a pre-defined orifice, at a pre-defined temperature, with a pre-defined load applied. It is intended to represent the actual injection molding process. Generally, the higher the melt index the easier the material will flow. However, melt index has a direct influence on material properties, with the lower melt index numbers exhibiting greater physical properties than the higher numbers for a given plastic.

Vicat Softening Point

The Vicat softening point is the temperature at which a small, heated probe (under light pressure) penetrates a specific distance into a specially-molded test specimen. It defines the ability of the plastic to resist short-term contact with a heated surface. The test is useful for crystalline materials but of limited value for amorphous materials which tend to creep during the test.

Heat Deflection (Distortion) Temperature

This is the temperature at which a plastic test bar, loaded to a specified bending stress, deflects by 0.010 in. (0.254 mm). This value is useful to the design engineer wishing to compare the relative ability of various plastics to perform at elevated temperatures while supporting loads. The value is listed as the temperature at which the flexural modulus decreases to particular values (i.e., 35,000 psi at 66 psi stress [241,309 kPa at 455 kPa]).

Coefficient of Linear Thermal Expansion

Plastic materials expand when heated and contract when cooled. The coefficient of linear thermal expansion, or CLTE, is the ratio of the change of a linear dimension to the original dimension, for a unit change of temperature. It is usually given in units of in./in./° F, or cm/cm/° C. This value is especially useful when assembling two different materials to determine how each will expand and contract when exposed to elevated temperatures during use. Be aware that many variables influence the CLTE and it may be different in a direction across the flow of plastic from a direction with the flow of plastic. It may also vary over a wide temperature range.

Thermal Conductivity

Thermal conductivity is the rate at which a plastic conducts heat along its length or through its thickness. It is important for the design engineer to understand this factor if the final product is used as a heat insulator.

Temperature Index

The temperature index is a value that must be stated if the product is to receive Underwriter Laboratories (UL) recognition, especially if it has electrical features. It defines the acceptable, maximum, continuous-use temperature for the product being recognized. The ratings are listed separately for electrical properties, mechanical properties without impact, and mechanical properties with impact.

Flammability

Flammability properties of plastic materials are valuable to the design engineer and end-user insofar as the consequences of exposing a plastic product to an open flame or other ignition source is concerned. This is a controversial subject, however, because most of the tests performed to determine flammability ratings must be conducted on specially-molded and conditioned test specimens rather than an actual molded part. Typical tests measure combustibility, smoke generation, and ignition temperatures. The two most common tests are the UL 94 Flammability Class and the Oxygen Index.

UL 94 Flammability Class (V-O, V-1, V-2, 5V, HB). The class identifiers in parentheses are used to determine the degree of flammability sought. The proper class is directed by the UL and is based primarily on the application of the final product. In this test, specimens are exposed to a specified flame. The ability to maintain combustion after the flame is removed becomes the basis for classification. Unfortunately, the classification is given based on tests performed on specially prepared test specimens and may not reflect the actual response of a molded product to a given situation. Each rating must include mention of the thickness of the test specimens used to determine the rating.

Oxygen Index. This test measures the percentage of oxygen necessary to sustain combustion of the plastic material being tested. Because the test can be performed on actual molded parts, it is considered more meaningful regarding the flammability concern for a specific plastic. The gas used is not pure oxygen but rather a mixture of oxygen and nitrogen starting at a ratio of approximately 20:80, respectively. Because normal air contains approximately 21% oxygen, any plastic with an oxygen index rating of less than 21 will support combustion in an open-air environment.

Electrical Properties

Plastic materials, by nature, are excellent electrical insulators and have found widespread use in a variety of electrical and electronic applications throughout

many industries. But besides insulation, there are many electrical properties that must be considered when using plastics for product designs. We cover the more important of these in this section. Test descriptions for determining electrical properties are defined in Chapter 9.

Volume Resistivity

Volume resistivity can be defined as the resistance of a plastic to an applied electrical current. It can be thought of as the inability of the plastic to conduct electricity. Volume resistivity is dependent upon thickness of the plastic being tested (specially prepared specimen) so the rating is generally given as ohm-cm. Materials with values above 10^8 ohm-cm are considered to be insulators, while those with values between 10^8 and 10^3 ohm-cm are considered to be partial conductors. Table V-1 shows typical values of some common thermoplastic materials.

Table V-1. Typical Volume Resistivity Values

Material	Volume Resistivity (ohm-cm)
Acetal	$10^{14} - 10^{16}$
Acrylic	$10^{16} - 10^{18}$
ABS	$10^{15} - 10^{17}$
Liquid crystal polymer	$10^{14} - 10^{17}$
Polyamide (nylon)	$10^{12} - 10^{16}$
Polyarylate	$10^{16} - 10^{17}$
Polycarbonate	$10^{15} - 10^{17}$
Polyester (TP)	$10^{14} - 10^{17}$
Polypropylene	$10^{14} - 10^{17}$
Polysulfone	$10^{15} - 10^{17}$
PPO (modified)	$10^{15} - 10^{17}$
PPE	$10^{15} - 10^{17}$
Polyphenylene sulfide	$10^{15} - 10^{17}$

Surface Resistivity

This property defines the ability of the plastic to resist a flow of electrical current across the surface of the plastic specimen. The rating of this property is influenced by surface contamination, especially moisture. The ratings can be used for comparing various groups of plastics for applications in which surface leakage may be a concern. Test data is collected under conditions that do not truly reflect the end-use environment, so large safety factors should be utilized by the design engineer.

Dielectric Strength

Dielectric strength is a measurement of the amount of voltage required to break down the insulation properties of a plastic specimen at a specific thickness. It is given as volts per millimeter and can be affected by temperature, conditioning of sample, and a variety of other factors, including contamination and internal voids, which may result in premature failure. It can be used as a determining factor in comparing groups of plastics for material selection purposes.

Dielectric Constant (Permittivity)

Dielectric constant is the value of a constant, a ratio between the permittivity of the plastic and the permittivity of air. Permittivity is best described as the ease with which plastic molecules can be polarized. Polarization occurs when a plastic material is exposed to an electrical field. The ratio that exists is based on air having a permittivity value of 1 (actually 1.000536). Table V-2 shows typical dielectric constants of some common thermoplastic materials.

Table V-2. Typical Dielectric Constant Values

Material	Dielectric Constant
Acetal	3.7 - 3.9
Acrylic	2.1 - 3.9
ABS	2.9 - 3.4
Liquid crystal polymer	3.7 - 10
Polyamide (nylon)	3.1 - 8.3
Polyarylate	2.6 - 3.1
Polycarbonate	2.9 - 3.8
Polyester (TP)	3.0 - 4.5
Polypropylene	2.3 - 2.9
Polysulfone	2.7 - 3.8
PPO (modified)	2.4 - 3.1
PPE	2.5 - 3.2
Polyphenylene sulfide	2.9 - 4.5

Dissipation factor. The dissipation factor is a measure of heat dissipation and is a constant ratio of the energy lost as heat compared to energy originally transmitted. The energy being transmitted is electrical energy and the heat is generated by the fluctuation of the polarization of plastic molecules. The dissipation factor is measured at 1 MHz (10⁶ cycles/sec). A low dissipation factor value is desired. Typical dissipation factors for common materials are shown in Table V-3.

Table V-3. Typical Dissipation Factor Values

Material	Dissipation Factor
Acetal	0.001 - 0.007
Acrylic	0.001 - 0.060
ABS	0.006 - 0.021
Liquid crystal polymer	0.010 - 0.060
Polyamide (nylon)	0.006 - 0.190
Polyarylate	0.001 - 0.022
Polycarbonate	0.0006 - 0.026
Polyester (TP)	0.0012 - 0.022
Polypropylene	0.003 - 0.014
Polysulfone	0.0008 - 0.009
PPO (modified)	0.0002 - 0.005
PPE	0.0002 - 0.005
Polyphenylene sulfide	0.001 - 0.002

Arc Resistance

Arc resistance is a time-rated value that measures the ability of a plastic material to withstand exposure to an electrical current across its surface before a conductive path of carbon is created. It is measured in seconds, and a high value is normally more desirable.

Comparative Tracking Index (CTI)

The CTI is similar to arc resistance except that an electrolyte (a solution of ammonium chloride) is placed on the surface of the plastic being tested. This better represents actual in-use conditions of a plastic part because of contamination that is normally present on products. The electrolyte reduces the arc resistance of the plastic because the electrical current tends to cross between electrodes along the electrolytic material.

Environmental Concerns

All materials are subject to environmental exposure. Metals rust and corrode. Plastics may crack, craze, discolor, and have their physical, mechanical, thermal, and electrical properties affected by the elements, as well as from exposure to chemicals, energy sources, or radiation. Some plastics are more resistant to various environments than others, as the following paragraphs describe.

Stress

Stress is created when the plastic molecules are forced into positions or areas in which they do not naturally wish to go. This can result from improper processing conditions during molding, improper product design (such as sharp corners), and even in assembly where one part can twist with another.

The degree of stress that occurs will determine the amount of resistance the plastic part has to its environment. In general, as the stress level increases, resistance to a particular environment decreases.

Exposure Factors

Exposure factors play a critical role in how an environment affects a plastic material. Some of these are:

- The length of *time* the plastic is exposed to an environment.
- The level of *concentration* of substance, chemical, or ingredient the plastic is exposed to in an environment.
- The *physical state* of the environment (solid, liquid, or gas).
- The amount of *radiation* present, as measured by level and intensity.
- Some *barriers*, such as certain paints or coatings, can be effective obstacles to intrusion by adverse environments.

Of course the *type* of environment has an effect on the plastic as well. Chemicals, weather, ultraviolet (UV) exposure, solvents—even water—can have detrimental effects on various plastics under certain conditions.

Temperature

Temperature can greatly influence the level of compatibility a plastic has with its immediate environment. The properties of amorphous plastics are affected almost immediately by any increase in ambient temperature, or in the temperature of a chemical on the surface. Crystalline materials tend to exhibit greater resistance to temperature increases due to their molecular makeup, but eventually they too are affected. The end-use and heat deflection temperature ratings can be used as indicators showing to what degree specific plastic groups react with the temperature of their environment.

Chemicals

The mechanism of chemical attack on plastics is complicated, and proper assessments are difficult. There are a variety of processes by which a chemical might attack a plastic. Some of the more common follow.

Reaction. In this mechanism the chemical attacks the polymer chain, causing a progressive lowering of the molecular weight of the plastic, resulting in lower properties, especially those considered to be mechanical or physical.

Solvation. Most thermoplastic materials can be dissolved in specific chemicals. The process is called solvation and is apparent through weight and dimensional changes, as well as swelling. The higher the molecular weight of a plastic, the slower the solvation process. Solvation causes a definite loss of properties, and the degree of dissolving is dependent on exposure time and temperature of the solvent. Table V-4 shows typical resistance of some common plastics to a variety of chemicals.

Table V-4. Chemical Resistance of Various Materials

<p>This information is presented for instructional purposes and is not intended for design. The data was extracted from numerous sources, making consistent rating assignments difficult. Further, the response of any given material to specific chemicals in any one class can vary significantly. Indeed, during the preparation of the table, the effect on one plastic of various chemicals in the same category ranged from essentially no effect to total dissolution. Therefore, an “A” rating for a particular plastic exposed to a particular class of chemicals should not be interpreted as applying to all chemicals in that class. The rating simply means that for the chemicals in that class found in the literature reviewed, the rating was generally an “A.” There may be other chemicals in the same class for which the rating would be a “C.” Finally, the typical chemicals listed do not necessarily correspond to the ones on which the individual ratings were based.</p>																		

Solvents																			
Aliphatic hydrocarbons	A	A	A	A	A	A	A	A	A	A	A	B	C	A	A	A	A	Heptane, hexane	
Aliphatic hydrocarbons, halogenated	A	B	C	B	B	A	A	A	C	C	C	—	—	—	B	B	B	Ethylene chloride, chloroform	
Alcohols	A	A	B	A	A	A	A	A	A	A	A	A	A	A	A	A	B	Ethanol, cyclohexanol	
Aldehydes	A	A	A	A	B	B	A	A	—	C	B	—	A	—	A	B	A	Acetaldehyde, formaldehyde	
Amines	—	—	—	—	—	—	C	B	—	C	C	—	A	—	A	B	B	Aniline, triethanolamine	
Aromatic hydrocarbons	A	B	A	A	B	B	A	A	C	C	C	C	C	C	A	A	A	Toluene, xylene, naphtha	
Aromatic hydrocarbons, halogenated	—	—	—	—	—	—	C	—	A	C	C	C	—	—	—	A	A	Chlorobenzene	
Aromatic, hydroxy	C	C	C	C	—	C	A	A	—	C	C	C	—	A	—	B	C	A	Phenol
Esters	B	B	A	B	B	B	A	A	C	C	C	C	—	C	—	B	B	B	Ethyl acetate, dioctyl phthalate
Ethers	B	—	A	A	—	—	—	A	—	A	B	—	C	—	A	A	A	Butyl ether, diethyl ether	
Ketones	B	B	A	B	B	B	A	A	C	C	C	—	B	C	A	A	A	Methyl ethyl ketone, acetone	
Miscellaneous																			
Detergents	A	—	A	—	B	—	—	A	A	A	—	B	A	—	A	A	B	Laundry and dishwashing detergents, soaps	
Inorganic salts	B	B	B	—	A	—	—	A	—	A	—	—	A	A	A	B	B	Zinc chloride, cupric sulfate	
Oxidizing agents, strong	C	C	C	—	C	—	B	B	—	C	—	—	A	—	C	C	C	30% Hydrogen peroxide, bromine (wet)	
Oxidizing agents, weak	C	C	C	A	—	A	A	A	—	A	—	A	A	A	B	C	A	Sodium hypochlorite solution	
Water, ambient	A	A	B	A	A	A	A	A	A	A	A	A	A	—	A	C	B		
Water, hot	B	C	B	C	C	B	A	A	—	C	—	A	C	—	A	C	B		
Water, steam	C	C	C	C	C	C	B	A	—	C	—	—	C	—	A	C	—		

A— Minimal effect

B— Some effect

C— Generally not recommended

*200° F (93° C)

Room temperature, except hot water, steam, and items denoted by asterisk.

Generally, extended exposure (more than a week) data was used.

(Courtesy Hoescht-Celanese Corp.)

Plasticization. If a chemical is miscible (soluble) with a plastic it may result in absorption and plasticization. Evidence of plasticization is indicated by a softening of the surface, reduction in tensile strength, reduced stiffness, an increase in impact strength, and a reduction of creep resistance. Swelling is also evident, and warpage may occur due to the relaxation of any molded-in stresses.

Stress cracking. The degree of chemical attack on a specific thermoplastic material is directly related to the level of stress in the plastic part. This stress (resulting from processing or product design issues) creates a product with less resistance to chemical attack. One phenomenon of this factor is that the same plastic may be affected in varying degrees in different products, because each product may have varying levels of stress. Actual chemical compatibility tests are performed on specially-molded and -conditioned unstressed samples rather than on a specific plastic product that may have high levels of stress. Therefore, chemical compatibility values are only useful for basic comparison among different plastic groups or similar plastics within a single group.

Weathering

Many thermoplastic material applications require that the end product be used in indoor areas with natural light or in outdoor areas exposed to sunlight. All thermoplastics are affected by weathering due to UV light exposure and degrade to varying levels. Such degradation includes fading, chalking, color change, and embrittlement.

Material suppliers have created UV light-resistant versions of most materials, but eventually all plastics will degrade when exposed to UV rays for long periods of time. Test procedures have been developed to determine the weathering characteristics (or weather resistance) of material test specimens, and these tests are usually performed in an accelerated state to determine the long-term effects of weather exposure.

SUMMARY

The variety of common properties associated with the performance of any thermoplastic material can be found in the categories of physical, mechanical, thermal, electrical, and environmental.

Physical properties include density, specific gravity, shrinkage rate, water absorption, transparency, modulus (toughness), elasticity, plasticity, ductility, brittleness, notch sensitivity, and lubricity.

Mechanical properties include stress and strain, modulus of elasticity, Poisson's ratio, shear stress and shear strain, shear modulus, direct shear, compressive strength and modulus, flexural strength and modulus, creep, impact strength, and fatigue endurance.

Thermal properties include melting point, glass transition temperature, melt index, Vicat softening point, heat deflection temperature, coefficient of linear thermal expansion, thermal conductivity, temperature index, and flammability (including Oxygen Index).

Electrical properties are those of volume resistivity, surface resistivity, dielectric strength, dielectric constant (permittivity), dissipation factor, arc resistance, and comparative tracking index.

Environmental concerns include stress, temperature exposure, chemical exposure, potential plasticization, stress cracking, and weathering.

Too much emphasis can be placed on *individual* mechanical properties of plastic materials, and while these can be used as indicators, a material's final performance is actually based on a *combination* of mechanical properties.

QUESTIONS

1. What are the five basic categories in which thermoplastic properties are listed?
2. Define density and how it is expressed.
3. Why are the terms "rate" and "shrinkage rate" misnomers?
4. How is the shrinkage factor usually expressed?
5. Define the term "constant" in Hooke's Law, and list other terms used to define it.
6. Define Poisson's ratio.
7. What is the formula for shear stress?
8. Define creep.
9. What are the four basic tests for determining the relative impact strength of a plastic material?
10. Of what use is the heat deflection temperature to a design engineer?

Applications and Properties of Specific Materials

6

CHARACTERISTICS AND APPLICATIONS

In this chapter we discuss typical descriptions and applications for some of the more common plastic materials available today. You will notice that some applications can use many of the plastics mentioned, and that some plastics can be used in various applications. This allows the product designer (or other person needing material selection knowledge) broad flexibility in material selection. With proper search information and correct design knowledge, the designer with such flexibility is able to create a product that can be produced at the lowest possible cost with the highest required level of quality.

Throughout the chapter, the materials are listed by generic name, alphabetically. In Appendix A you will find a tradename reference that briefly describes the generic nature of a variety of materials under their common tradenames. Appendix B lists plastic material acronyms and their meanings. In appendix C, plastic materials are ranked according to values of specific properties.

Common Thermoplastics

Acrylonitrile-butadiene-styrene (ABS)

One of the major engineering plastics used today, ABS is an amorphous material known for its outstanding impact strength, reasonable cost, and relative ease of processing. It also exhibits good dimensional stability, temperature resistance above the boiling point of water, chemical resistance, and electrical properties. It is available in a wide range of grades and impact strengths. ABS is commonly blended with other materials, such as PVC, to provide higher impact, lower cost, inherent flammability resistance, and color stability.

Typical of the products for which ABS is specified are television and electronic enclosures, automobile dashboard components, industrial carriers and trays, knobs, safety helmets, refrigerator and appliance parts, and carrying handles.

Acetal

Acetal is a crystalline material and one of the more rigid thermoplastics. It is known for its ease of machining and resistance to most chemical solvents, though it cannot be made flame retardant. It has a low coefficient of friction, and good

toughness, fatigue life, and tensile strength. It exhibits good creep resistance and dimensional stability (although high shrinkage), but has a relatively high specific gravity.

Typical acetal products include gears, bushings, bearings, plumbing components, carburetors and fuel contact components, door handles, and appliance parts.

Acrylic

Acrylics are amorphous materials normally used in applications requiring a transparent plastic. Acrylic materials do not have inherent high-impact strength properties, but special high-impact grades are available. This family of plastics maintains valuable optical properties and exhibits maximum colorability. Many copolymers, or alloys formed with other monomers, are available. Acrylics also effectively withstand weathering, making them particularly suited to outdoor products. These materials have excellent scratch and abrasion resistance.

Acrylics are typically specified for products such as window panes, lighting fixtures, lenses (including automotive, binocular, camera, contact, and instrument), piano keys, display cabinets, signs, and textile fibers.

Acrylic-styrene-acrylonitrile (ASA)

ASA materials are amorphous plastics, known for their weatherability. They are often alloyed or blended with other polymers such as PVC, PBT, or polycarbonate. Physical properties such as processability (including the need to thoroughly dry the material prior to use) are similar to those of ABS.

Unpainted exterior auto parts, pools and spas, outdoor furniture, signs, lawn and garden equipment, marine accessories, and siding, are typical of the products made from ASA.

Cellulose

Cellulosic materials are one part of a large family consisting of five cellulose variations named *nitrate*, *acetate*, *butyrate*, *propionate*, and *ethyl cellulose*. They are basically crystalline materials, although amorphous versions are available.

While *nitrate* is the toughest member of the cellulose family, it is extremely flammable and highly explosive, difficult to process, and becomes brittle with age. It was the first of the synthetic plastics and is the material the Hyatt brothers used to win their contest. Its low flash point is the main reason it is not in common use today. Typical products made from nitrate include combs, brushes, buttons, collar stays, and photographic films.

Acetate is not explosive, is much less flammable than nitrate, takes brilliant color, is transparent and easy to process, but exhibits poor solvent resistance and becomes brittle with age. It was first used in photographic plates and gained popularity as a thread for the synthetic fabric industry. Its value lies in its toughness,

colorability, and fairly high-impact strength. Typical products made from acetate include beads, knobs, various toys, sunglass frames, packaging materials, and magnetic tapes.

Butyrate (actually called cellulose acetate butyrate) materials were developed to counter the poor moisture resistance and dimensional stability of acetate. They are similar to acetates, but are used for applications requiring improved resistance to moisture, as well as better dimensional stability. Butyrates are known for their toughness and ability to maintain brilliant colors. Typical products made from butyrate include tool handles, steering wheels, street lamp globes, outdoor signs, blister packaging, and metallized sheet and film.

Propionate materials are known for their short cycling requirements and freedom from lamination defects during molding. They exhibit better weathering properties than acetate, have high impact resistance and toughness, and much less of a tendency to become brittle with age. Typical products made from propionate include toys, pen barrels and caps, various automotive parts, toothbrushes, steering wheels, handles, and small electronics cases.

Ethyl cellulose has very high shock resistance even at low temperatures (240° F [240° C]), but has poor weather resistance. It has higher impact strength and heat distortion temperatures than any of the other cellulose materials and is free of odor. Ethyl cellulose materials are used for various military products (because of their freedom from low temperature embrittlement), decorative trim, toys, cosmetic cases, tool handles, and blister packaging.

Fluoroplastics

The four basic types of fluoroplastics—TFE (polytetrafluoroethylene), FCTFE or CTFE (polychlorotrifluoroethylene), PFEP or FEP (polyfluoroethylene-propylene), and PVF (polyvinylidene fluoride)—are fundamentally crystalline materials. They have high chemical resistance, very low coefficient of friction, low moisture absorption, and exhibit excellent dimensional stability. TFE has very high heat and chemical resistance (up to 500° F [260° C]), excellent resistance to heat aging, and is nonburning, but is very difficult to process. CTFE has most of the same properties as TFE but is somewhat easier to mold. FEP is very easy to mold, and has most of the properties of TFE, but is the most expensive fluoroplastic. PVF is the easiest to mold of all fluoroplastics, but has heat resistance only to 300° F (149° C). However, it exhibits higher tensile strength and lower cold flow than the other fluoroplastics.

Typical fluoroplastic products include cable jacketing, coil forms, wire insulation, nonstick surfaces, O-rings, chemical apparatus and laboratory ware, and tubing.

Ionomers

Ionomers are crystalline polymers composed of ionized hydroxyl groups. They are most commonly used in sheet or film versions, but are available in molding

grades. They exhibit excellent toughness and high optical clarity, high abrasion resistance down to -300°F (-184°C), outstanding flexibility down to -160°F (-107°C), high melt strength, excellent electrical properties, and resistance to stress cracking. They are limited to applications whose ambient temperatures are below 170°F (77°C).

Typical products made of ionomers include packaging, film laminates, and coextruded food containers.

Liquid Crystal Polymers

These materials are a unique class of crystalline engineering thermoplastics whose main features are thin section stiffness, extreme temperature stability, and excellent chemical resistance. They exhibit outstanding electrical properties, superior resistance to most chemicals, and because of excellent flow characteristics, are readily processed.

Typical products made of liquid crystal polymers include a variety of electrical connectors and components, and consumer cookware.

Methyl Pentenes

Methyl pentenes are crystalline materials, primarily produced by the Mitsubishi Chemical Company of Japan. They are commonly known as TPX resins and are similar to polypropylene in physical properties and processability, except that they have much better high-heat resistance (up to 430°F [221°C] short-term). They have electrical properties similar to TFE, a very low specific gravity (less than 0.85), good chemical resistance, and excellent transparency.

Typical products made of methyl pentenes include medical bottles, disposable syringes, clear cooking pouches, cosmetic containers, disposable packaging and containers, and transparent pipe and tubing.

Nylon (Polyamide)

Many different types of polyamide resins are available today including 4/6, 6, 6/6, 6/9, 6/10, 6/11, 6/12, and an aromatic polyamide. They are all crystalline materials except for type 12, which is also available as an amorphous resin. In fact, there are over 100 different types and combinations of polyamides, all noted for their varying levels of toughness, abrasion resistance, strength, low friction, and high heat resistance. They are self-extinguishing, and each of the individual types have different degrees of toughness, stiffness, and moisture absorption properties. Nylon materials resist all common organic solvents, except phenol and formic acids. Strong mineral acids and oxidizing agents affect them, but only slowly. Glass reinforcement is commonly added to increase these properties. One major drawback is that nylons are hygroscopic (absorbing moisture from atmosphere) and thus have poor dimensional stability.

Typical polyamide products include gears, cams, bearings, shoe heels, wire insulation, bottles, fuel lines, gunstocks, and various fabric and filament applications.

Polyallomers

These crystalline materials lie somewhere between polyethylene and polypropylene, and can be considered a combination of both. They are comparable with high-density polyethylene in many respects, and can be used at temperatures ranging from -40 to 210°F (-40 to 99°C). Polyallomers combine rigidity with high impact resistance and have very high stress-crack resistance. They are used when extreme “living” hinge technology is required because they are superior to polyethylene and polypropylene for this characteristic.

Polycarbonates

Polycarbonates are amorphous materials that exhibit excellent heat resistance, high impact strength, good dimensional stability, and excellent transparency. Although they are self-extinguishing, they have only fair chemical resistance. They are soluble, or partially soluble, in aromatic and chlorinated hydrocarbons. Polycarbonate resins do not stain, tarnish, or corrode, making them ideal for food containers.

Typical products made from polycarbonates include street light globes, safety “glass” panes, armor shields, power tool housings, appliance components, popcorn makers, and hot-dish handles.

Polyetheretherketone (PEEK)

PEEK materials are crystalline, and are known for their thermal stability, with short-term applications up to the 600°F (316°C) range and long-term uses to approximately 550°F (288°C), when glass-reinforced. These materials are capable of meeting UL 94V-0 flammability requirements without flame retardant additives. PEEK demonstrates excellent fatigue and abrasion resistance and long-term load-bearing ability.

Typical products made from PEEK include electrical terminal blocks, electrical connectors, printed and 3D molded circuit boards, manifolds, aircraft components, and sterilized medical products.

Polyethylene

Polyethylene is one of the resins from the polyolefin family. It is a crystalline material known for its excellent chemical resistance to most acids, bases, and salts. It is available in a wide variety of densities ranging from “low” (often called regular or branched) to “high” (often called linear), and even “ultrahigh.” Although it is considered a “commodity” resin, as are most in the polyolefin family, and is inexpensive, it does exhibit properties that make it valuable for use in the laboratory (for containers) or when a slippery surface or nonstick surface is required.

Typical polyethylene products include chemical apparatus, laboratory ware, detergent and oil bottles, milk containers, plastic grocery bags, packaging film,

squeeze bottles, surgical implants, wire and cable insulation, pipe fittings, and a variety of disposable items.

Polyimides

Polyimides are available as both amorphous and crystalline materials. They are extremely difficult to process owing to their high thermal requirements and heat resistance. They maintain continuous-use heat resistance to 500° F (260° C), and up to 900° F (482° C) for short periods. They have a very low coefficient of expansion, high impact strength, and excellent wear resistance. They are well known for their electrical properties. Due to the difficulty in processing, the products molded of polyimide are normally sintered.

Typical polyimide products include bearings, compressor valves, piston rings, and diamond-abrasive wheel binders.

Polyphenylene Oxide (PPO)

Polyphenylene oxide plastics are amorphous resins from a family of materials that have a useful temperature range from -275 to 375° F (-170 to 191° C). They are nontoxic, have very low moisture absorption, excellent creep resistance, and are self-extinguishing. In addition, they exhibit excellent chemical resistance and dimensional stability. The normal grades are expensive and difficult to mold. However, a modified version is available that molds easily and costs less, but has a lower heat resistance.

Typical applications for polyphenylene oxide are products such as autoclavable surgical tools, pump housings, various electrical insulator devices, coil housings and forms, pipe, and pipe fittings.

Polyphenylene Sulfide (PPS)

PPS is basically a crystalline material that exhibits high temperature resistance as well as high physical properties through a wide temperature range. It is non-flammable and has excellent chemical resistance to all known solvents below 400° F (204°C). Its electrical properties are outstanding, and it is easy to process, though very hot molds are required, which usually means that oil heaters will be needed.

Typical products made of PPS include electrical connectors, electronic housings, chemical pumps and filters, bearings, and various high-temperature application components, both industrial and automotive.

Polypropylene

Another of the polyolefin family, polypropylene, a crystalline material, is known for its excellent resistance to stress cracking or flex cracking, thus its popularity for living hinges. It has a very low specific gravity, excellent impact strength, a scratch-resistant surface, good chemical and heat resistance, and exhibits

excellent moisture barrier properties. It is inexpensive, known as a commodity resin, and very easy to process.

Typical products made of polypropylene include washing machine agitators, various housewares and appliances, automotive ducts and trim, washer and dishwasher components, hinged containers and cases, bottles, cabinets, and packaging.

Polystyrene

Polystyrene is an inexpensive, easily processed, amorphous material available in a variety of grades, and is a popular resin for alloying various other resins, such as ABS. The basic polymer is characterized by its rigidity, sparkling clarity, and ease of processing, but tends to be brittle, with low impact strength. Modified polystyrenes are produced by copolymerizing with elastomers to increase impact resistance, resulting in materials known as high-impact, and even super high-impact polystyrenes. They range from very rigid to rubbery in nature. All polystyrenes have poor weatherability and have low resistance to chemical attack by oils and organic solvents. However, they have good resistance to inorganic chemicals, water, and alcohols.

Typical products made of polystyrene include plastic glassware, eating utensils, storage containers, toys, model kits, knobs, see-through lids and covers, and low-cost lenses.

Polysulfones

Polysulfones are amorphous materials that offer excellent high-temperature and minimum creep properties for load-bearing and structural applications requiring long-term use at temperatures up to 300° F (149° C). Although processing can be tricky (with barrel temperatures up to 750° F [399° C]), the molded products are resistant to mineral acids, alkalis, salt solutions, most detergents, oils, and alcohols. This material is damaged by ketones, chlorinated hydrocarbons, and aromatic hydrocarbons. Polysulfones exhibit excellent electrical properties, are available in a variety of colors, and are resistant to stress cracking.

Typical products made of polysulfones include under-the-hood automotive components, hot water pipes, iron handles, dishwasher and pump impellers, switches, circuit breakers, and electronic hardware.

Polyvinylchloride (PVC)

PVC is a family of amorphous materials available in two basic forms: rigid (as for pipes, etc.) and flexible (simply called vinyl).

Their wide range of available properties allow them to be used in a variety of applications requiring chemical resistance and outdoor exposure. Because of their inert sensitivity to processing temperatures they are easily degraded during molding and care must be exercised to maintain proper temperatures and consistent parameter settings.

Typical products made of PVC include household siding, plumbing pipes and pipe fittings, dashboard panels and covers, and business machine covers.

Thermoplastic Polyesters (PBT, PET)

The two most common thermoplastic polyesters are polybutylene terephthalate (PBT) and polyethylene terephthalate (PET). The polyesters are crystalline materials. PBT crystallizes rapidly and is an excellent material for fast cycling and for products requiring good electrical and mechanical properties. PBT exhibits outstanding resistance to stress, cracks, and a wide variety of chemicals. Filled versions retain their properties up to 425° F (218° C).

Typical PBT products include electrical connectors, insulators, chemical containers, valves and housings for chemical apparatus, and a variety of automotive components.

PET, owing to its unusual molecular orientation capability, produces products with very high strength properties. It is commonly used for film and fiber, but not limited to those areas. PET materials exhibit a limited crystallinity temperature range, so processing windows are narrow. Complete and thorough drying prior to processing is absolutely essential. PET plastics are used for such products as carbonated beverage bottles, blownware preforms, and electrical connectors. In its film version it is used for screens, thread, magnetic tape, cooking bags, and packaging.

Thermoplastic Copolyesters (PCTA, PCTG, PETG)

Thermoplastic copolyesters are chemically different from the polyesters owing to the incorporation of more than one glycol or dibasic acid. They exhibit a much reduced tendency to crystallize. Depending on the basic processing procedures, some are amorphous and some are crystalline. Processing procedures are critical and proper drying is essential. Typical products made from copolyesters include beverage containers, bottles, and packaging.

Polyarylate (Wholly Aromatic Polyester) (ARP)

Aromatic polyesters can be either amorphous or crystalline. They are tough, transparent, impact resistant, and exhibit good weathering resistance. Their most important feature is their inherent high flame resistance (UL 94V-0) without the need for flame-retardant additives. In addition, they have excellent electrical properties and a heat deflection temperature of 345° F (174° C) (at 264 psi [1820 kPa]). They do exhibit hydrolytic instability, similar to the polycarbonates, but show good stability at up to 300° F (149° C) under relatively high humidity.

Typical products made of ARP include electrical connectors and insulators, signs, laboratory and electronic cabinetry and housings, and kitchenware.

Thermoplastic Rubbers (TPR)

TPRs differ slightly from natural or synthetic rubbers in that they can be processed without compounding or including additives. They have excellent resilience, toughness, and chemical resistance, and can be processed by a variety of methods. Typical products made of TPR include off-road vehicle tires, poppet valve seats, ski shoes, auto body components, gaskets, toys, and sporting goods.

Elastomers

The term *elastomers* includes a wide spectrum of rubber-like polymers that exhibit varying degrees of elasticity. They are also referred to as synthetic rubbers, and sometimes, simply rubbers. Because such a large number of rubber-like polymers now exist, the term elastomer is more fitting and is used most commonly. A molded elastomer, by definition, must meet two important criteria:

- Capability of being stretched 100%,
- Capability of retracting to within 10% of its original length within 5 minutes after being stretched 100% and held for 5 minutes.

Descriptions in the following paragraphs include common names, standard American Society for Testing and Materials (ASTM) nomenclature, basic chemical descriptions, and general properties for the more popular elastomers available today.

Natural Rubber (NR)

Most natural rubber (also referred to as latex) comes from the *Hevea* tree. It is chemically described as isoprene in a natural state, and polyisoprene when various resins are combined. Natural rubbers exhibit low hysteresis, high resilience, and very low water absorption.

Isoprene Rubber (IR)

Isoprene rubber is the true synthetic equivalent of natural rubber. Whereas natural rubber must be sorted, washed, screened, dried, blended, and masticated prior to processing, isoprene synthetics are ready to process and offer purity and controlled molecular weights for uniform processing. They are generally described chemically as polyisoprenes and have the same general properties as NR but with fewer impurities and better water resistance.

Styrene-butadiene Rubber (SBR)

Commonly known as buna-S, and chemically described as a styrene-butadiene copolymer, SBR is designed for general purpose use. It has better heat resistance than NR but is not as rubbery. It is known as the original rubber substitute for making tires.

Neoprene (CR)

Considered the first commercial synthetic rubber, neoprene is chemically described as polychloroprene. It is a general-purpose, oil-resistant, weather-resistant synthetic, widely used in cover insulation. It is self-extinguishing and ideal for applications where fire is a hazard.

Nitrile Rubber (NBR)

Also known commonly as buna-N, NBR is chemically described as a butadiene-acrylonitrile copolymer. It is a general-purpose material but provides better oil resistance than neoprene. It exhibits very good general-purpose chemical resistance and is physically similar to SBR, but has better heat resistance.

Butyl Rubber (IIR)

Butyl rubber is chemically known as an isobutylene-isoprene copolymer. It exhibits very low air permeability, good weathering, and good electrical properties. It is also known for its excellent energy absorbing qualities. It maintains flexibility at low temperatures and has very good insulation properties.

Ethylene Rubber (EPM)

Ethylene rubber is chemically described as an ethylene-propylene copolymer. Generally it is similar to butyl rubber but has higher air permeation and better heat resistance. It is inexpensive, and has excellent weathering properties, but is not oil resistant.

Ethylene Terpolymer Rubber (EPDM)

Very similar to the EPM copolymer, EPDM is chemically described as a terpolymer, consisting of ethylene, propylene, and a side chain of diene. It is inexpensive, exhibits the same properties as EPM but is faster curing, and has become the standard general-purpose rubber for nonoil applications.

Hypalon™ (CSM)

Hypalon is a trade name product developed by DuPont Chemical Corp. and chemically described as a chlorosulfonated polyethylene. It is hard to distinguish from neoprene for properties, but it has better color stability and improved chemical and heat resistance. It is commonly used for spark plug boots.

Acrylic Rubber (ABR)

Chemically described as acrylate butadiene, ABR is similar to NBR but has much better hot oil resistance. It is not as good as NBR for low temperature applications, but is not affected by automotive oils, and therefore commonly is used for transmission fluid seals.

Polysulfide Rubber (T)

Polysulfide rubber is chemically described as organic polysulfide. It has good weather resistance and solvent resistance but emits an objectionable odor. It has

lower properties than NBR in many areas, but because of its weatherability and solvent resistance, is commonly used for outdoor caulking, printing press rolls, and solvent-carrying hoses.

Silicones (SI, FSI, PSI, VSI, PVSI)

The silicone rubbers are chemically described as poly-dimethyl siloxanes, with many variations. Rather than the carbon structure of other elastomers, these materials have a silicone and oxygen backbone structure. Their most outstanding properties are heat resistance (to 500° F [260° C]) and flexibility at low temperatures (below -100° F [-73° C]). They are often used in electrical insulation applications because, if decomposed, they still act as an insulator, whereas nonsilicone materials decompose to form conductive carbon black. The basic silicone materials are modified with groups, such as vinyl or fluoride, which act to enhance properties of tear and oil resistance and other physical and environmental characteristics.

Urethanes (U)

Urethanes are chemically described as reaction products of diisocyanates and polyalkylene ether glycols. They have high tensile strength, wide hardness range, high abrasion resistance, and are extremely tough. They process like polyvinyl chloride. They have excellent resistance to oils, solvents, oxidation, and ozone, but poor resistance to hot water (over 175° F [79° C]). They are especially useful when load-bearing and wear properties are required.

Fluoroelastomers (FPM)

Depending on the specific compound, the fluoroelastomers are chemically described as either copolymers of vinylidene fluoride and hexafluoropropylene, or copolymers of chlorotrifluoroethylene and vinylidene fluoride. They exhibit outstanding chemical solvent resistance and excellent heat resistance to 450° F [232° C]). They have excellent weathering characteristics, but are very expensive.

Butadiene Rubbers (BR)

Chemically described as polybutadiene, these materials are mostly used as additives to improve the wear properties of SBR products, such as tires. Because they are very difficult to process in themselves, they are seldom used at more than a 60:40 compound ratio with other polymers. These materials exhibit excellent abrasion resistance, and their other properties are very similar to SBR.

Carboxylic (COX)

Chemically described as butadiene-acrylonitrile modified with carboxylic groups, these materials are similar to NBR but with improved ozone resistance and superior cold-temperature properties. Their outstanding abrasion resistance (due to vulcanization) approaches that of urethane.

Epichlorohydrin Rubbers (CO, ECO)

These materials are chemically described as either homopolymers of epichlorohydrin, or copolymers of epichlorohydrin with ethylene oxide. They exhibit the combined properties of both NBR and CR. Recognized as having excellent resistance to swelling in oils and fuels, they are also resistant to acids, bases, and water.

STAR DIAGRAMS

Figures 6-1 through 6-26 (pages 108 through 133) are “star” diagrams describing the properties of a variety of common plastic materials. A star diagram is a rating system, with a long “ray” indicating that the material rates well on that point. The information was supplied by *Plastics World* magazine and is reprinted with their permission. These diagrams are designed to assist material selection personnel by exhibiting pertinent material information at a glance. The information covers a variety of particulars specific to each resin. Additional information can be obtained by contacting a material supplier directly.

STRUCTURAL FOAMS

While the foam process is not really within the scope of this book, the use of structural foams has been increasing over the last 10 years, and they are now considered a viable option to standard injection molding materials. The list of foamed materials has also grown steadily with a typical roster including polyurethanes, polycarbonates, polyphenylene oxide, phenylene ether copolymers, polybutylene-terephthalate, ABS, polyethylene, polystyrene, and polypropylene. These materials are used for molding products such as business machine housings, fascias, medical and electronic cabinetry, and material handling equipment. By far the largest use is in furniture applications where the foamed materials have the look, feel, machinability, and appearance of natural wood. Materials suppliers are valuable sources for information on characteristics and applications of specific structural foams.

The common processes for molding structural foam parts are reaction injection molding (RIM), gas counter-pressure, injection molding using in-mold foaming agents, injection molding using two-part resin systems, injection molding using precompounded foaming resins, and the relatively new co-injection process.

Use of structural foam materials provide the product designer and manufacturer several advantages, such as the ability to mold large detailed parts, low production costs, rigidity, capability to form complex shapes and varying wall thicknesses, high strength-to-weight ratio, low density, low pressure requirements, minimal or no sink marks, low molded-in stress, improved chemical

resistance, and low tooling costs. Low tooling costs derive from the low pressures required to fill the mold, thereby allowing aluminum molds and other “soft” mold materials.

Some of the disadvantages of structural foams are that their surfaces usually require secondary finishing or painting, there is a sacrifice of physical properties compared to nonfoamed base resin, and a possible requirement of somewhat longer cycle times than standard injection molded parts, owing to the increased wall thicknesses and time needed to facilitate the foaming activity.

If you are interested in more information on structural foam materials and processes, contact any of the larger material suppliers. They will be glad to share resource information with you.

SUMMARY

Common thermoplastic materials come in a variety of types with a variety of properties and processing ranges. In addition, these materials are available over a wide range of prices.

The number of available materials (over 20,000) allows flexibility for the product designer and person responsible for selecting the right material for a specific application.

With proper search information and correct design knowledge, a person is capable of selecting the proper material for producing a plastic product at the lowest possible cost with the highest required level of quality.

Star diagrams provide a visual representation of the strong and weak points of a specific plastic and show at a glance how one material compares to another.

Structural foam materials have grown in popularity, generating interest by the material suppliers to provide a wider variety of foam materials.

QUESTIONS

1. What should be the result if a person with correct design knowledge and proper search information performs a material selection?
2. What do the letters ABS stand for concerning plastic materials?
3. What is the advantage to using a star diagram for material selection?
4. What is acetal most noted for?
5. What are the four basic types of fluoroplastics?
6. Which of the fluoroplastics is easiest to mold?
7. What are the main features of liquid crystal polymers?
8. What is another common name for *polyamide*?
9. What is the useful temperature range for polyphenylene oxide materials?
10. How do thermoplastic rubbers differ from natural or synthetic rubbers?

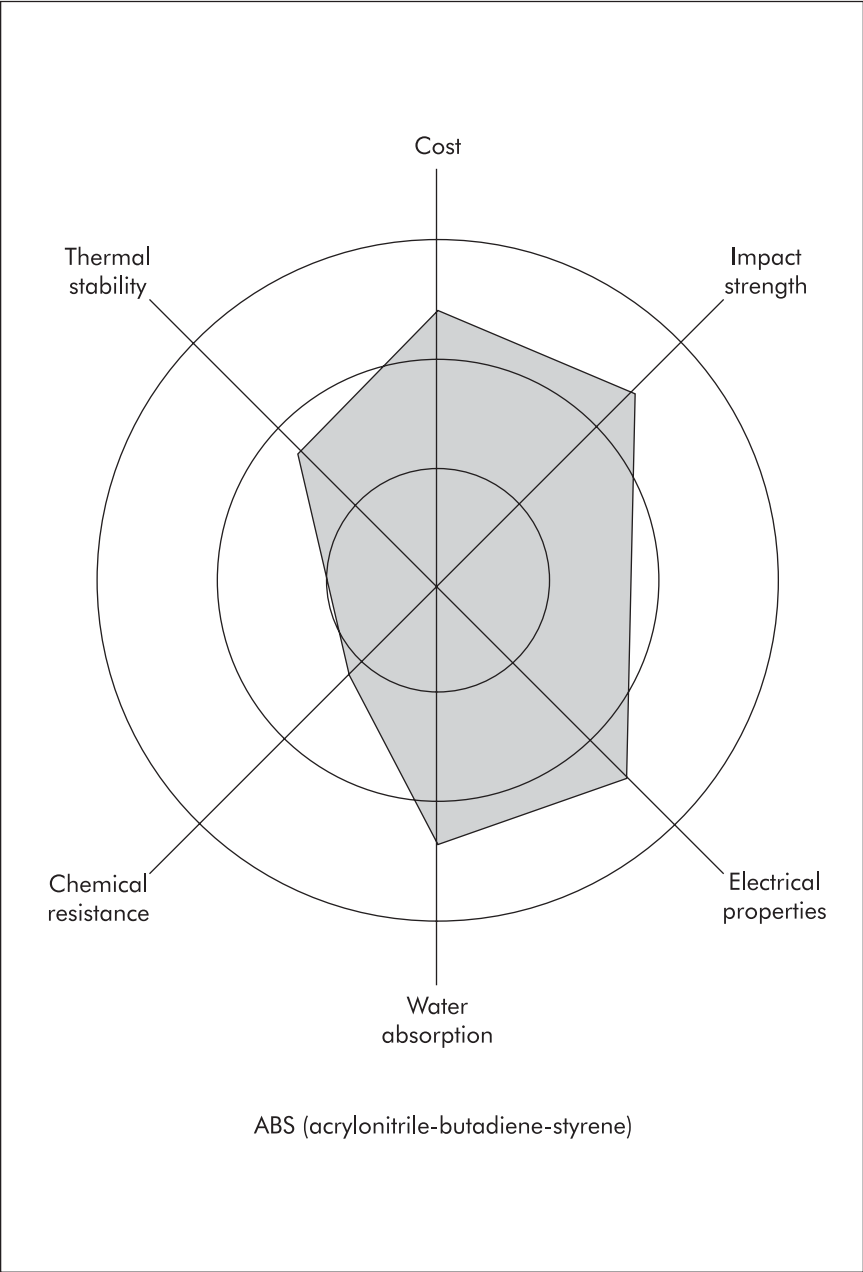


Figure 6-1. Acrylonitrile-butadiene-styrene (ABS) is an amorphous engineering thermoplastic whose main features are impact resistance, rigidity, high gloss, and low cost relative to other engineering plastics.

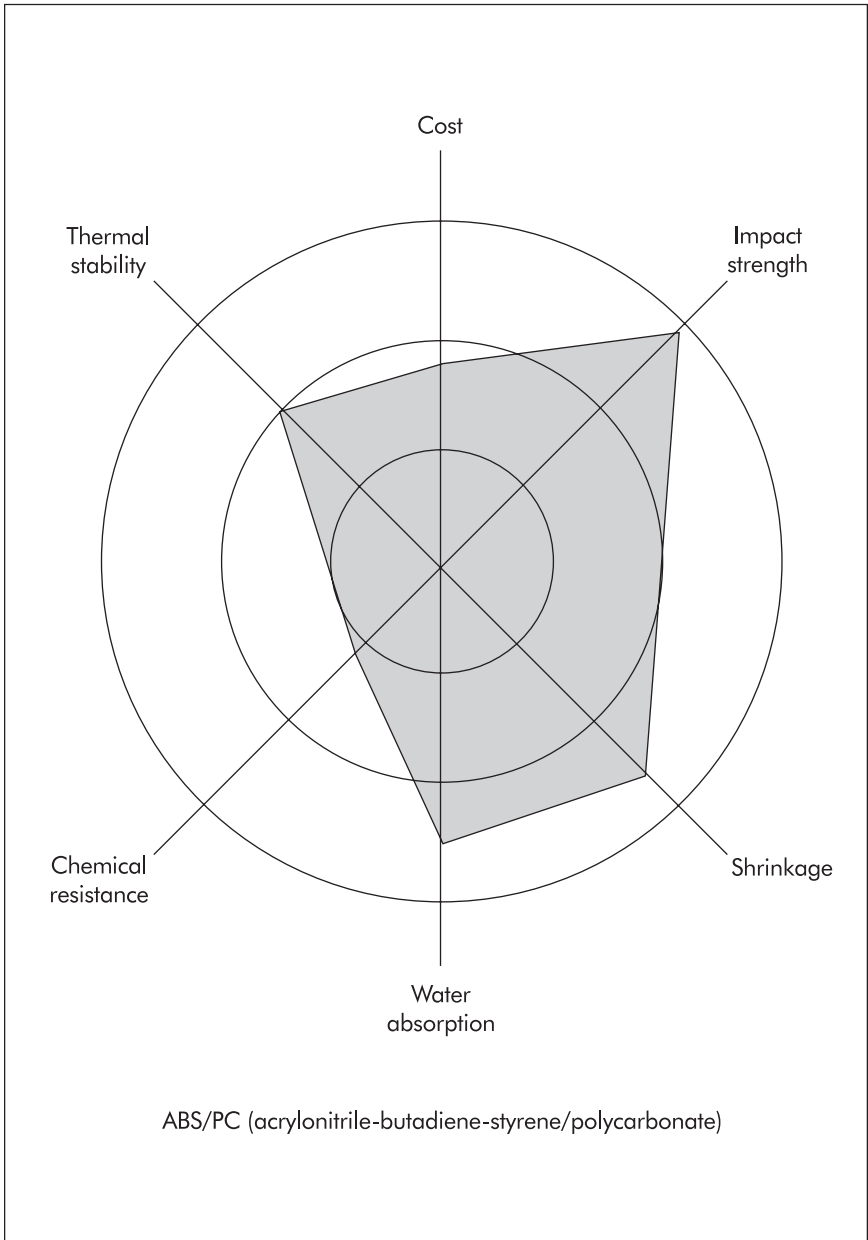


Figure 6-2. ABS/PC resins are blends of the amorphous engineering thermoplastics, acrylonitrile-butadiene-styrene (ABS), and polycarbonate (PC). Together, these yield materials with good impact resistance over a wide temperature range, rigidity, dimensional stability, and good creep resistance.

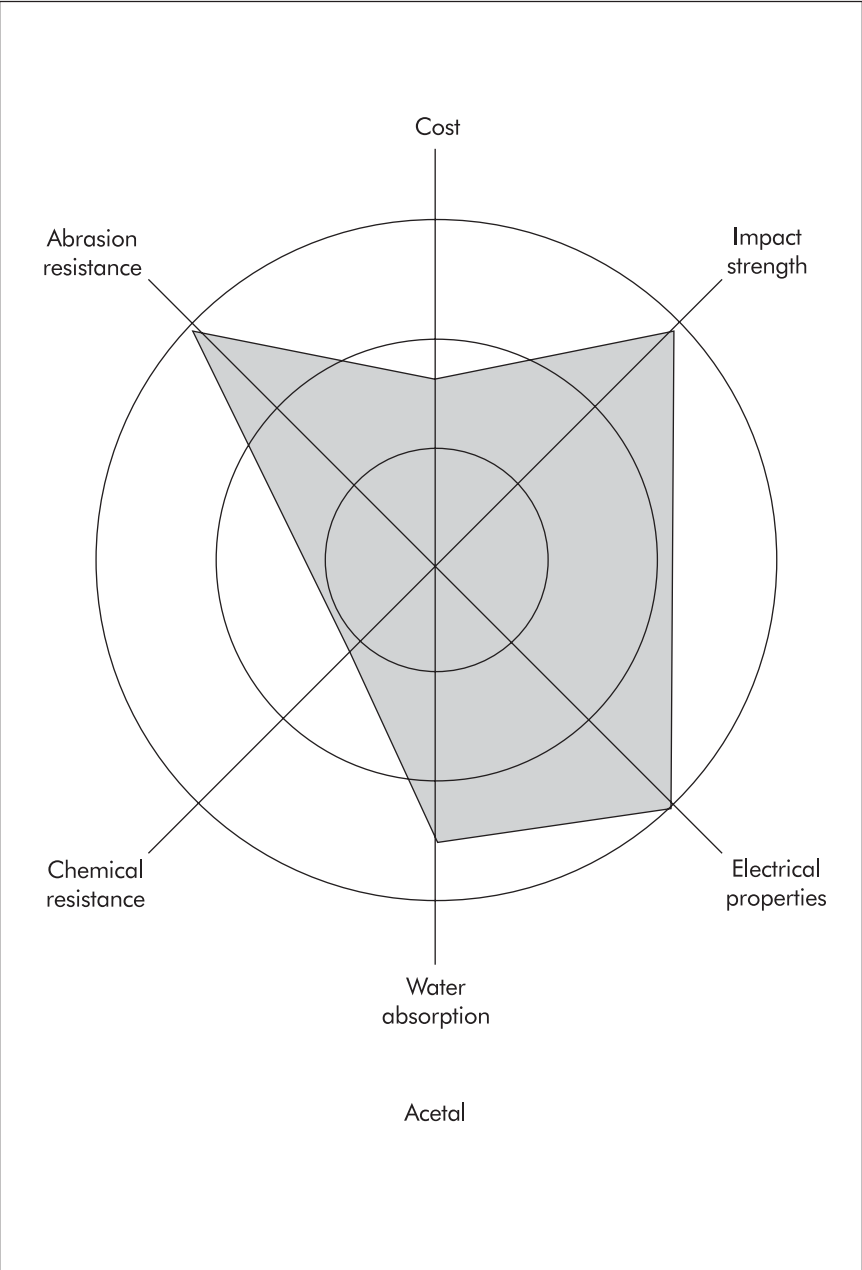


Figure 6-3. Acetal resins are crystalline engineering thermoplastics whose main features are stiffness, creep resistance, low coefficient of friction, fatigue endurance, and good appearance.

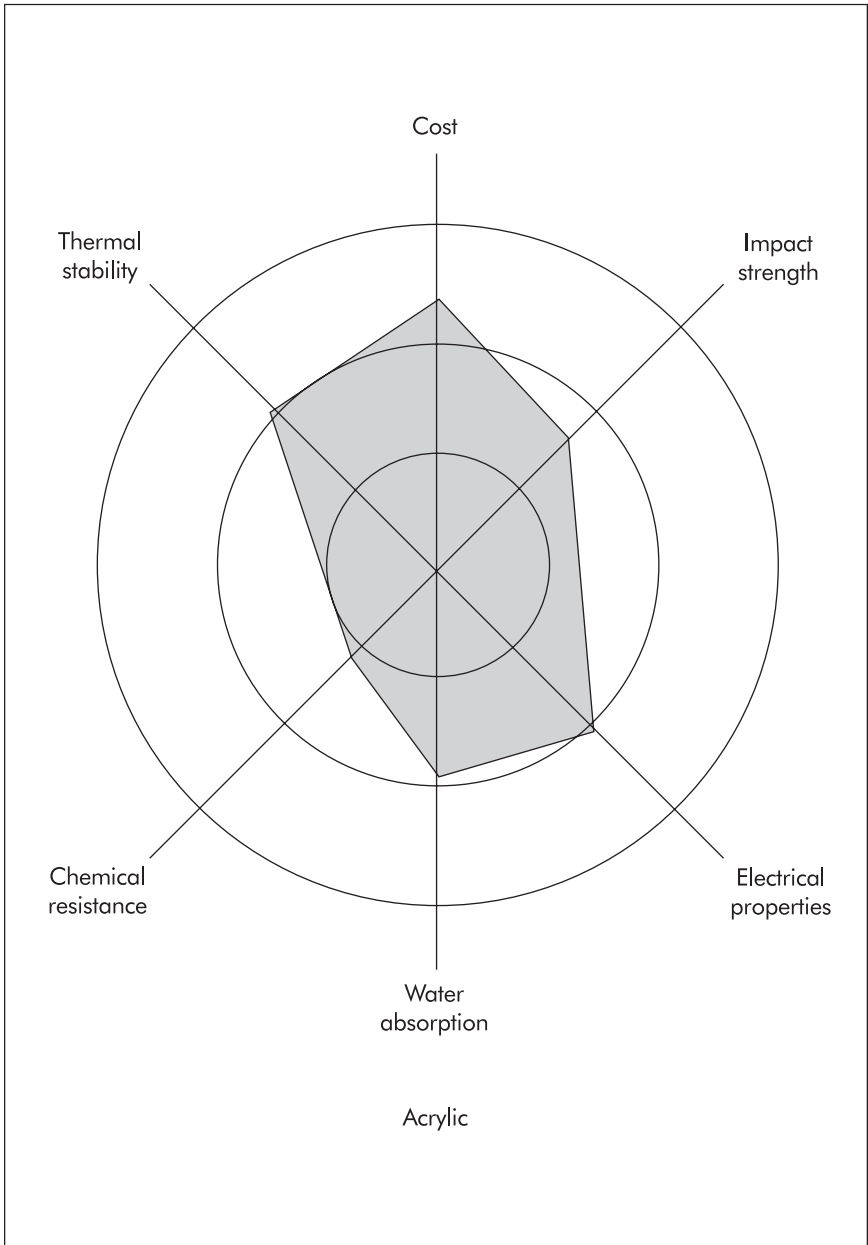


Figure 6-4. Acrylics are amorphous thermoplastics featuring exceptional optical properties and good weather resistance, along with basically good strength, electrical, and chemical resistance properties. They do not discolor after molding and post-mold shrinkage is minimal to nonexistent.

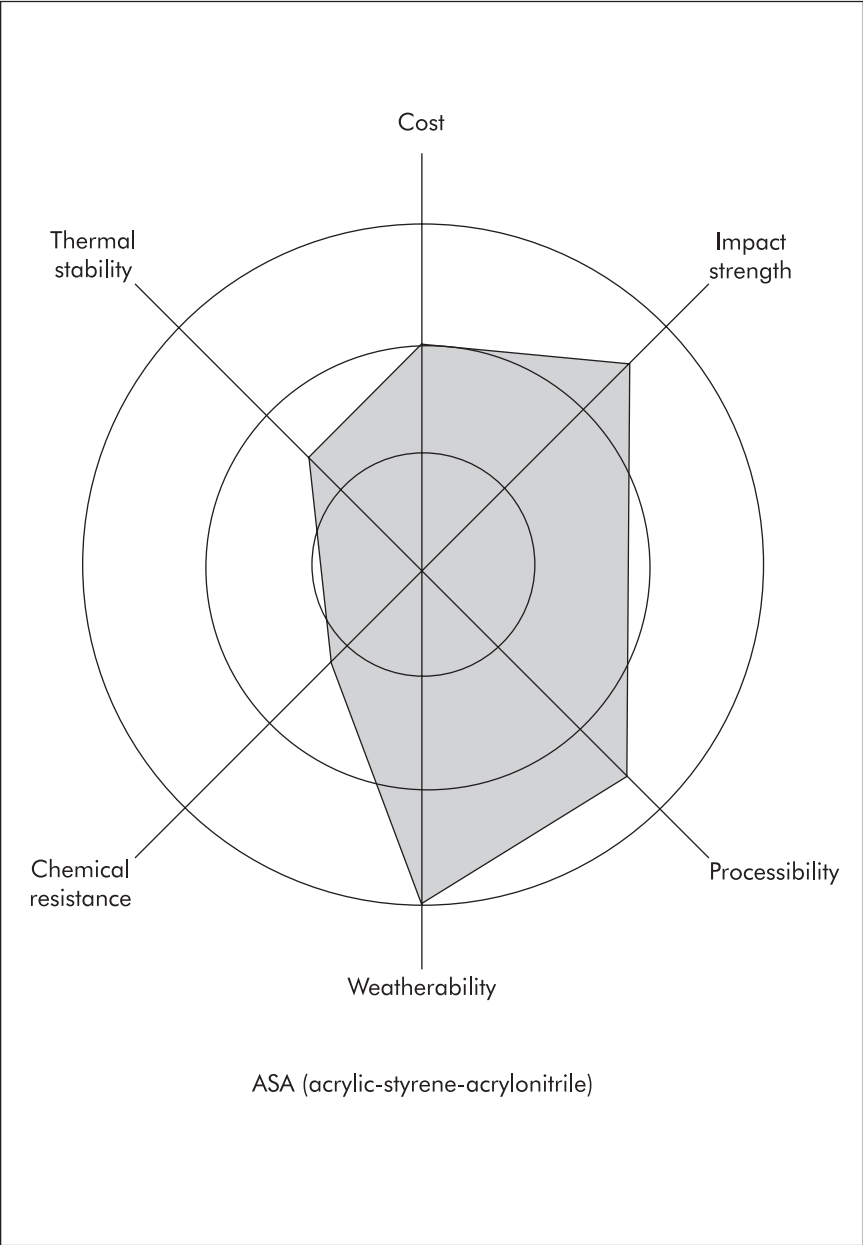


Figure 6-5. Acrylic-styrene-acrylonitrile (ASA) resins—amorphous polymers prized for their weatherability—feature mechanical properties similar to ABS polymers. ASA materials can be alloyed or blended with other polymers such as PVC or polycarbonate to produce resins with combined features of both basic resins.

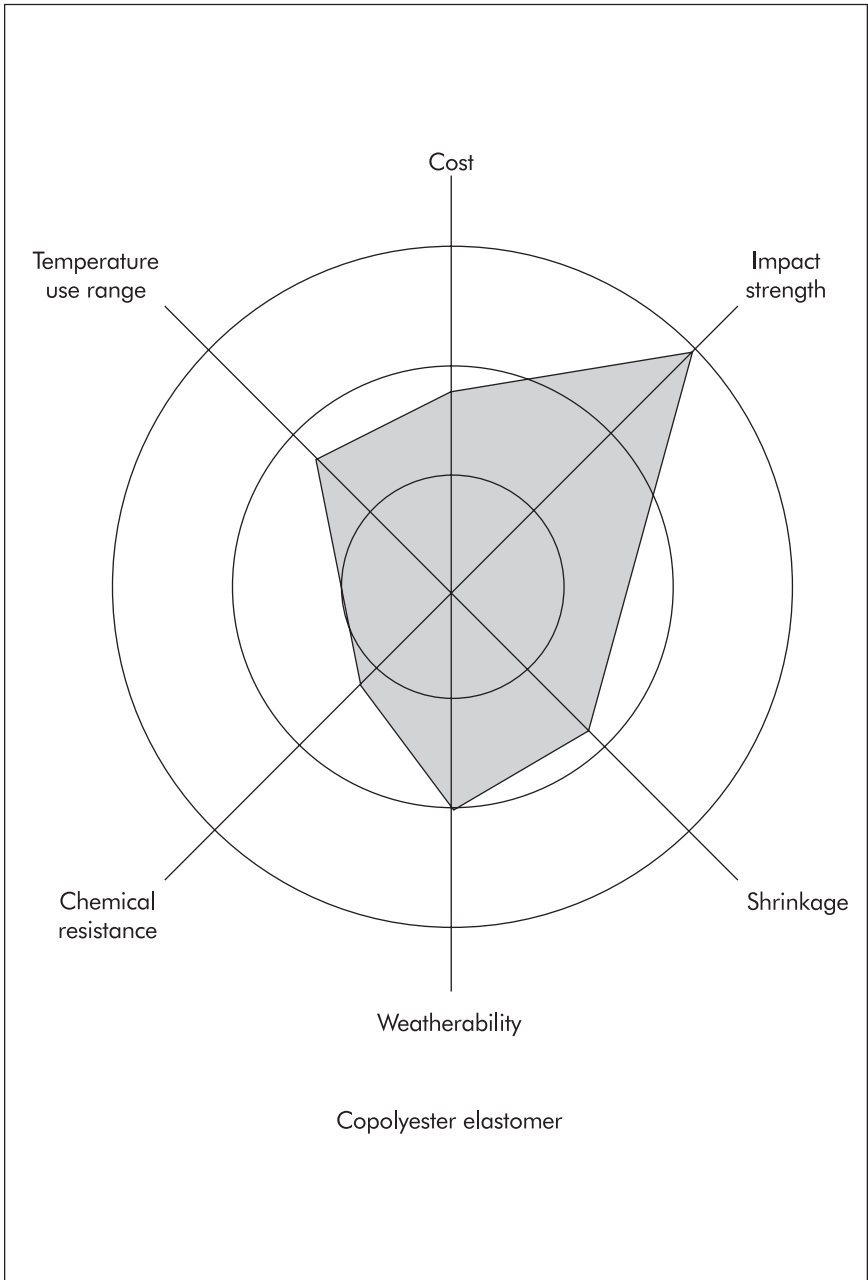


Figure 6-6. Copolyester elastomers (COPE) are semicrystalline thermoplastics incorporating wide temperature range flexibility, impact strength, chemical and weathering resistance, tear strength, and abrasion resistance.

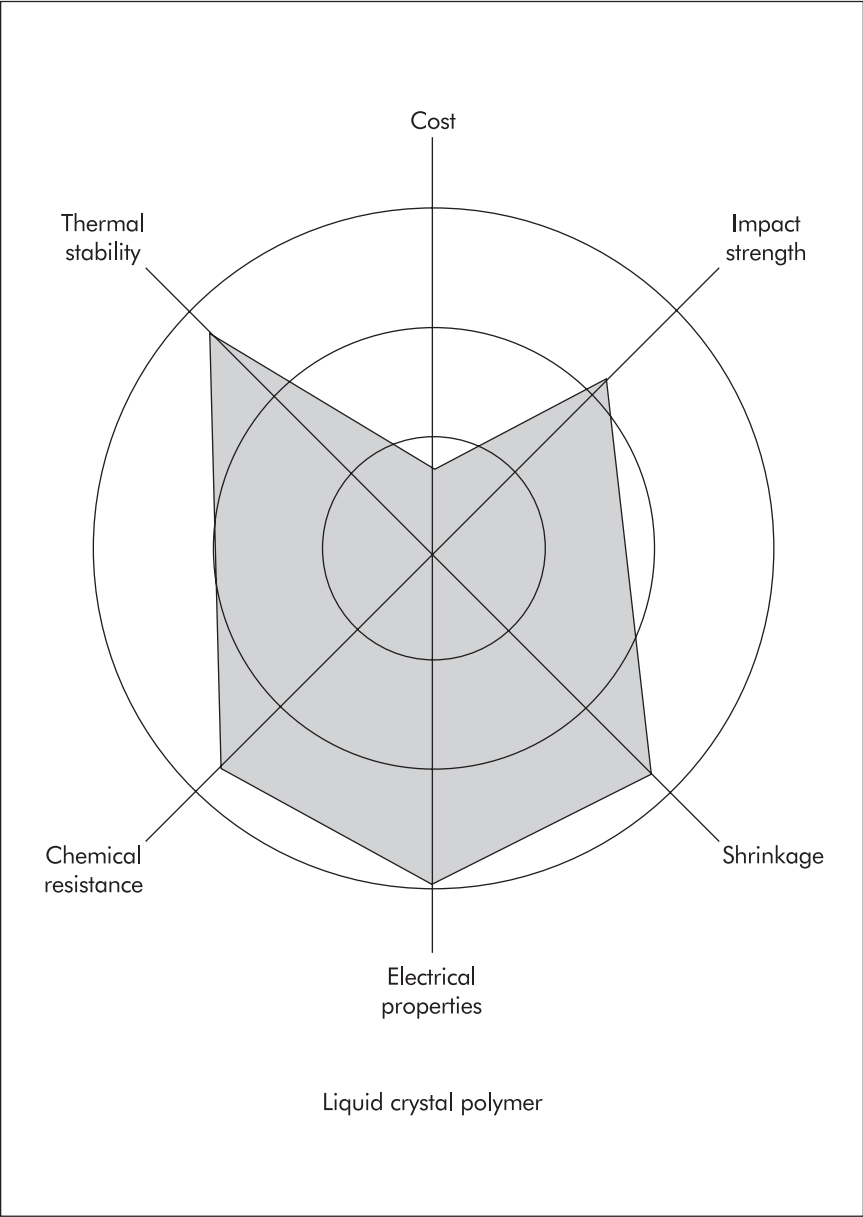


Figure 6-7. Liquid crystal polymers (LCPs) are a unique class of crystalline engineering thermoplastics whose main property strengths are high stiffness in thin sections, extreme temperature stability, and good chemical resistance. While early versions were difficult to process, advances in formulations have greatly improved the processability of these materials.

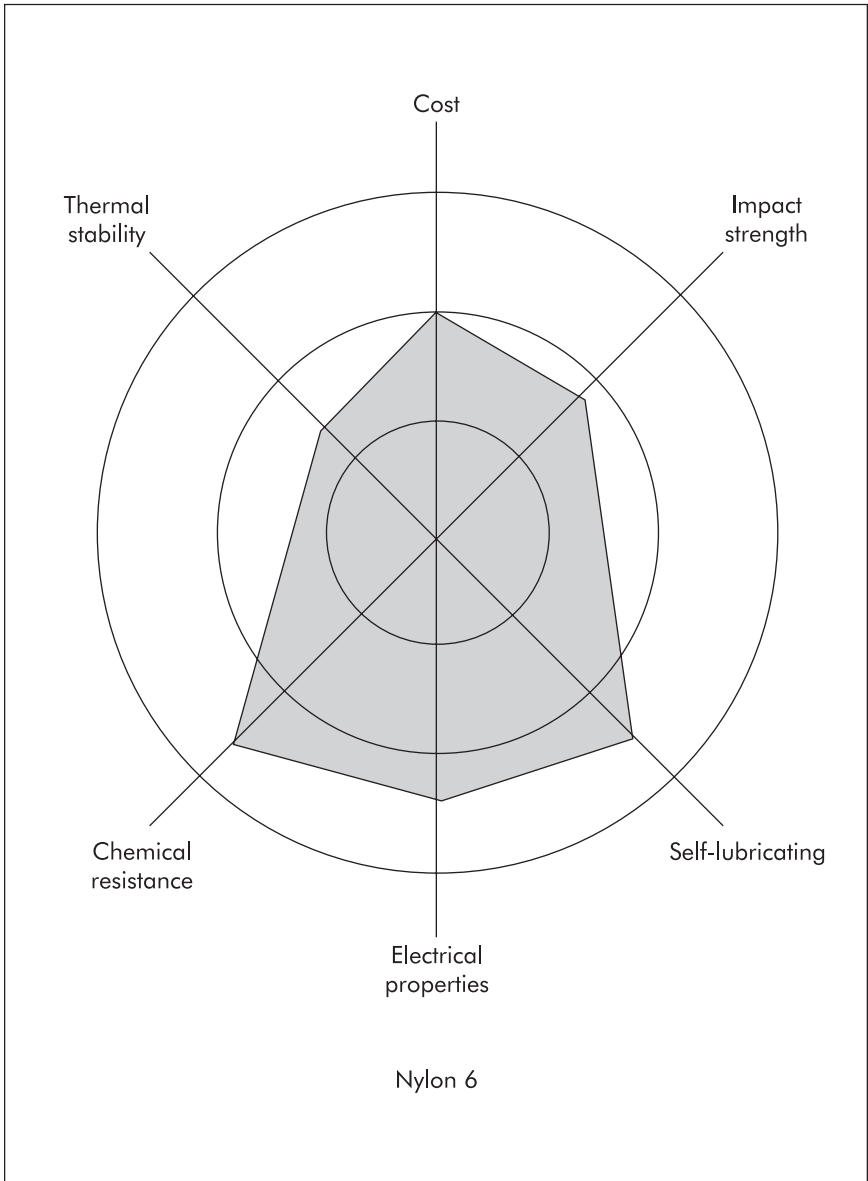


Figure 6-8. Nylon 6 is a semicrystalline polyamide thermoplastic valued for its strength, toughness, and chemical resistance. It is similar in properties and performance (but with a lower melting point) to its close cousin, nylon 6/6 and is used in many of the same applications. Like other polyamides, nylon 6 is hygroscopic (absorbing moisture from the atmosphere) which decreases stiffness and tensile strength, but increases toughness.

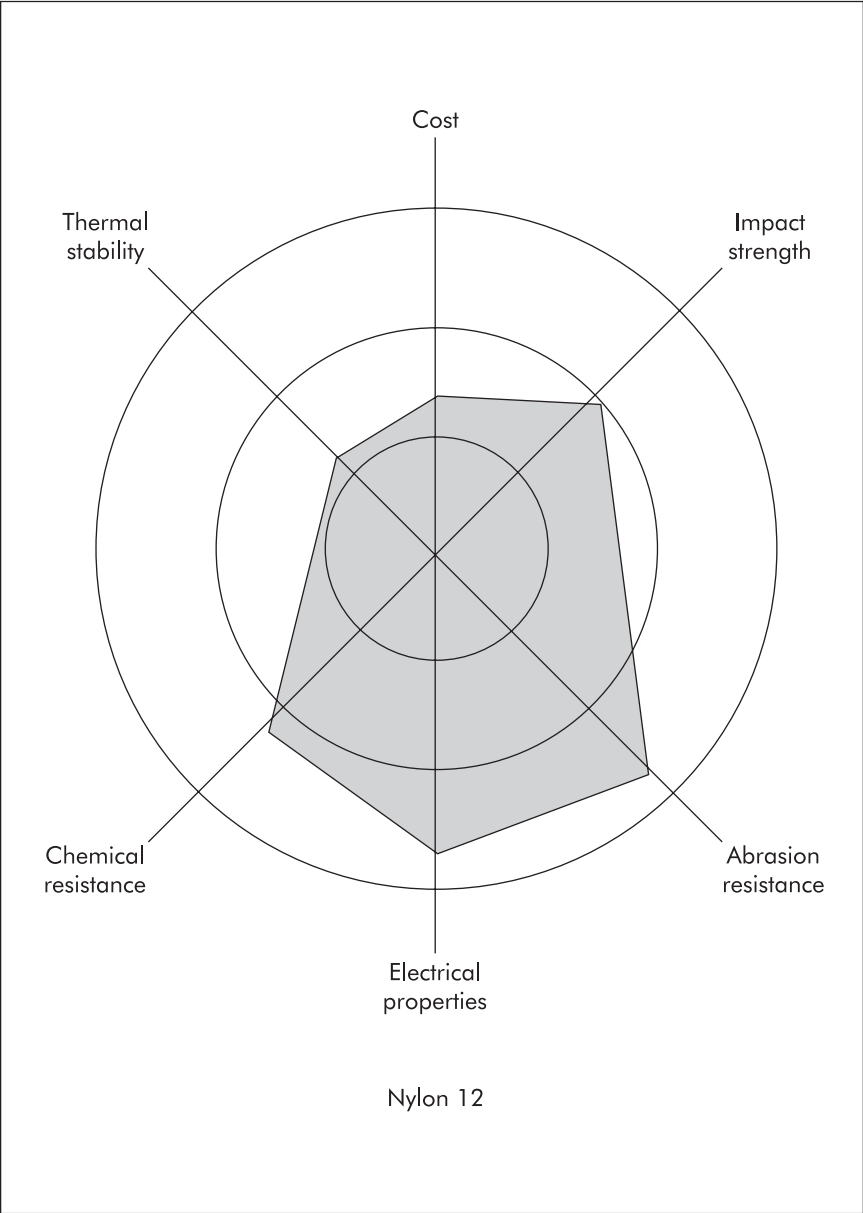


Figure 6-9. Nylon 12 is a crystalline engineering thermoplastic whose main features are resistance to chemicals, friction, and abrasion, as well as impressive mechanical and electrical properties. Nylon 12 is similar to other nylons in terms of fatigue and impact resistance, low friction, and resistance to aromatic hydrocarbons, but absorbs much less moisture than nylons 6 and 6/6.

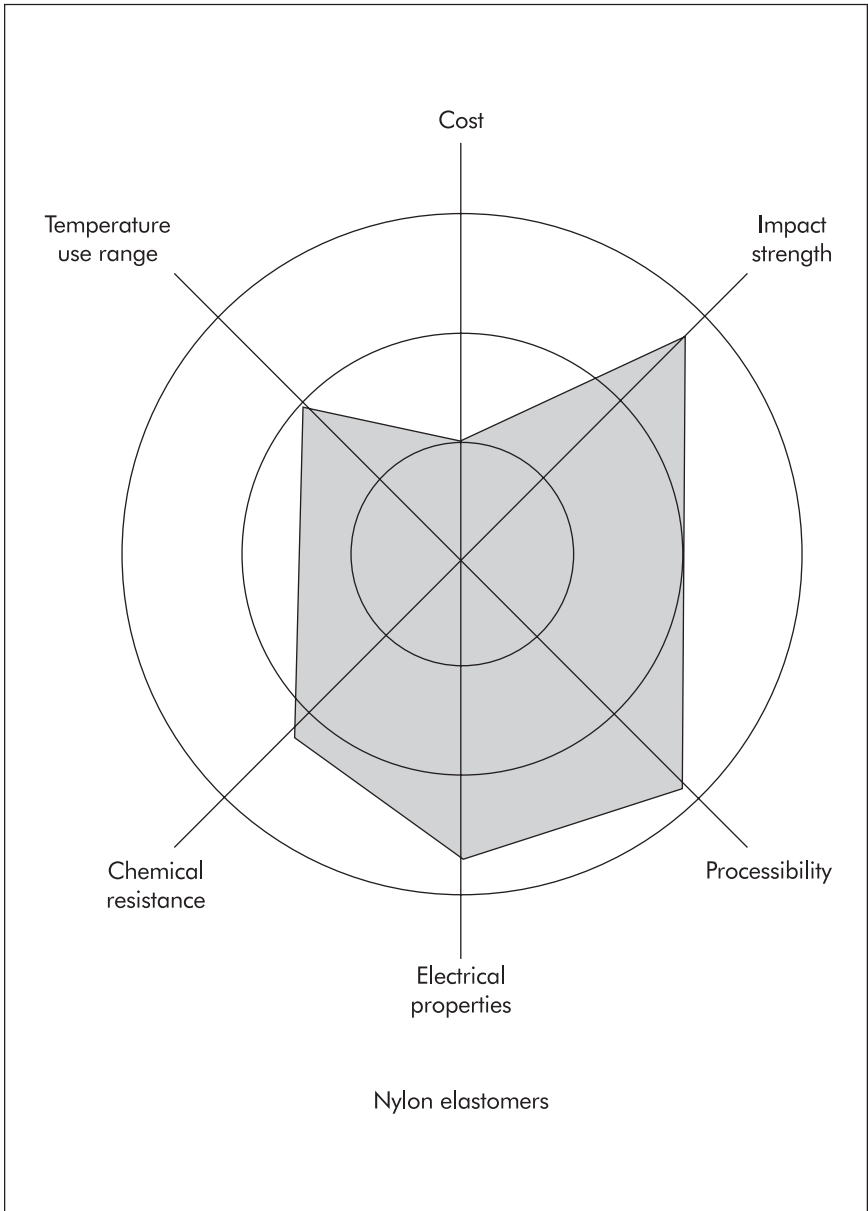


Figure 6-10. Nylon (polyamide) elastomers are thermoplastic materials made up of rigid crystalline polyamide segments connected by flexible, amorphous polyether, polyester, or poly-etherester segments. Like thermoplastic urethane (TPU) elastomers, these high-end materials are tough and wear resistant over a wide temperature range.

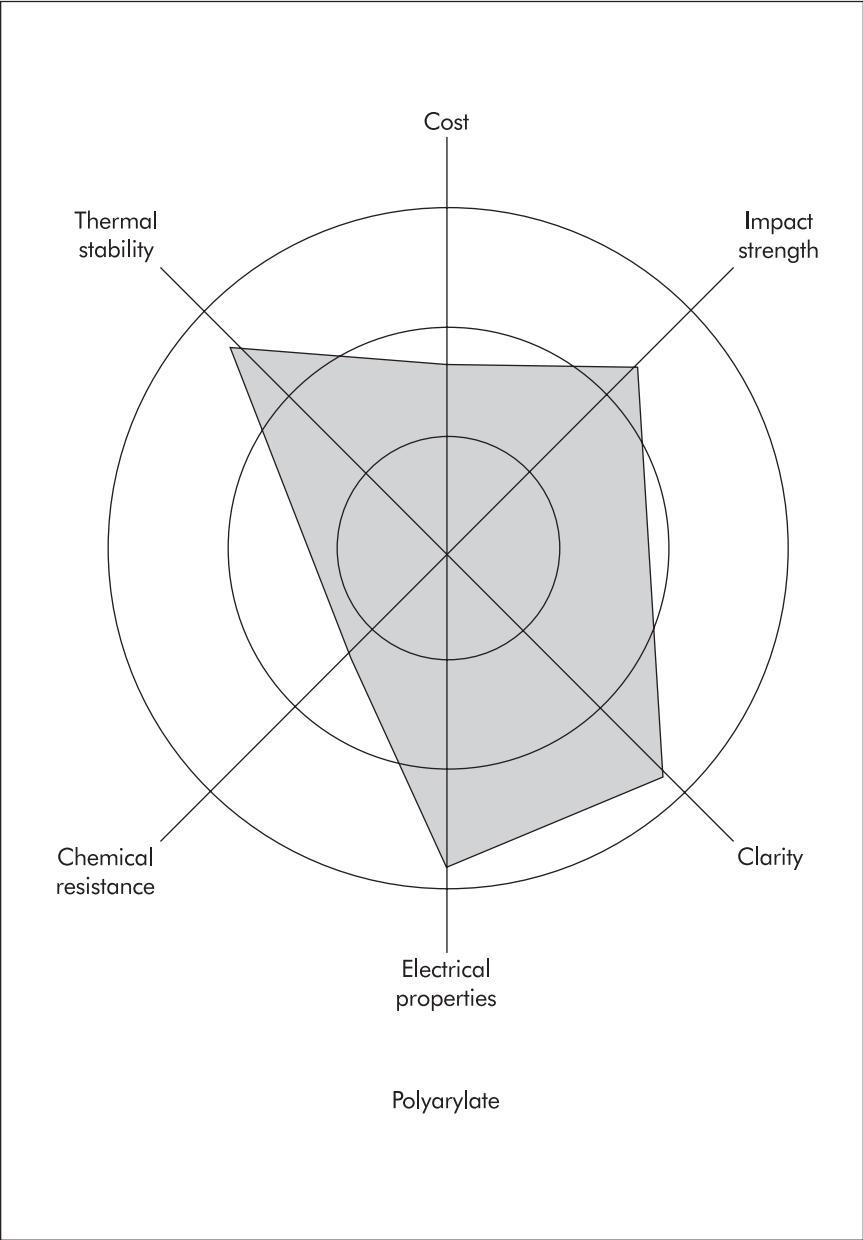


Figure 6-11. Polyarylates are a family of amorphous engineering thermoplastics known principally for their heat resistance, impact strength, inherent flame retardancy, good weatherability, and clarity. They also exhibit excellent electrical properties.

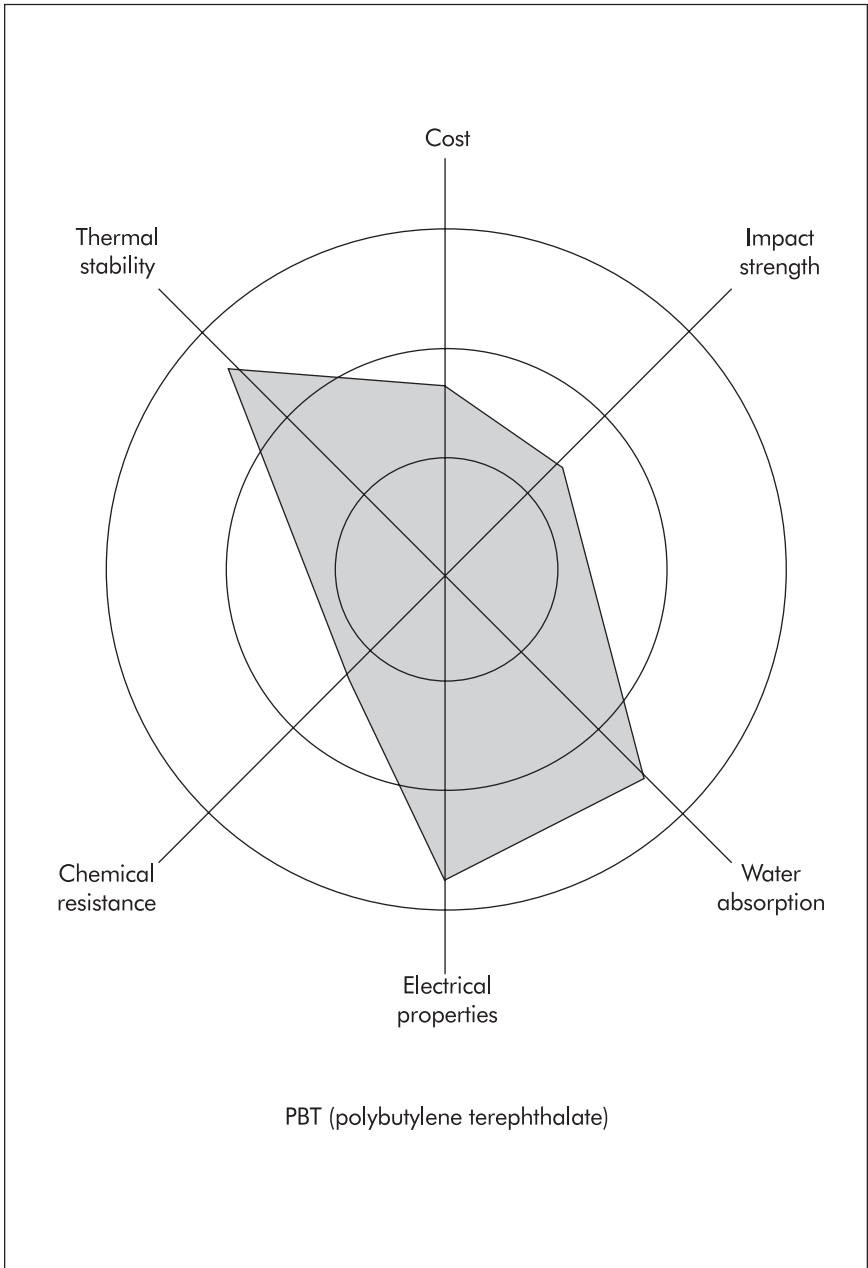


Figure 6-12. Polybutylene terephthalate (PBT) is a crystalline engineering thermoplastic used in products needing stability under load, low surface friction, low water absorption, and good electrical properties.

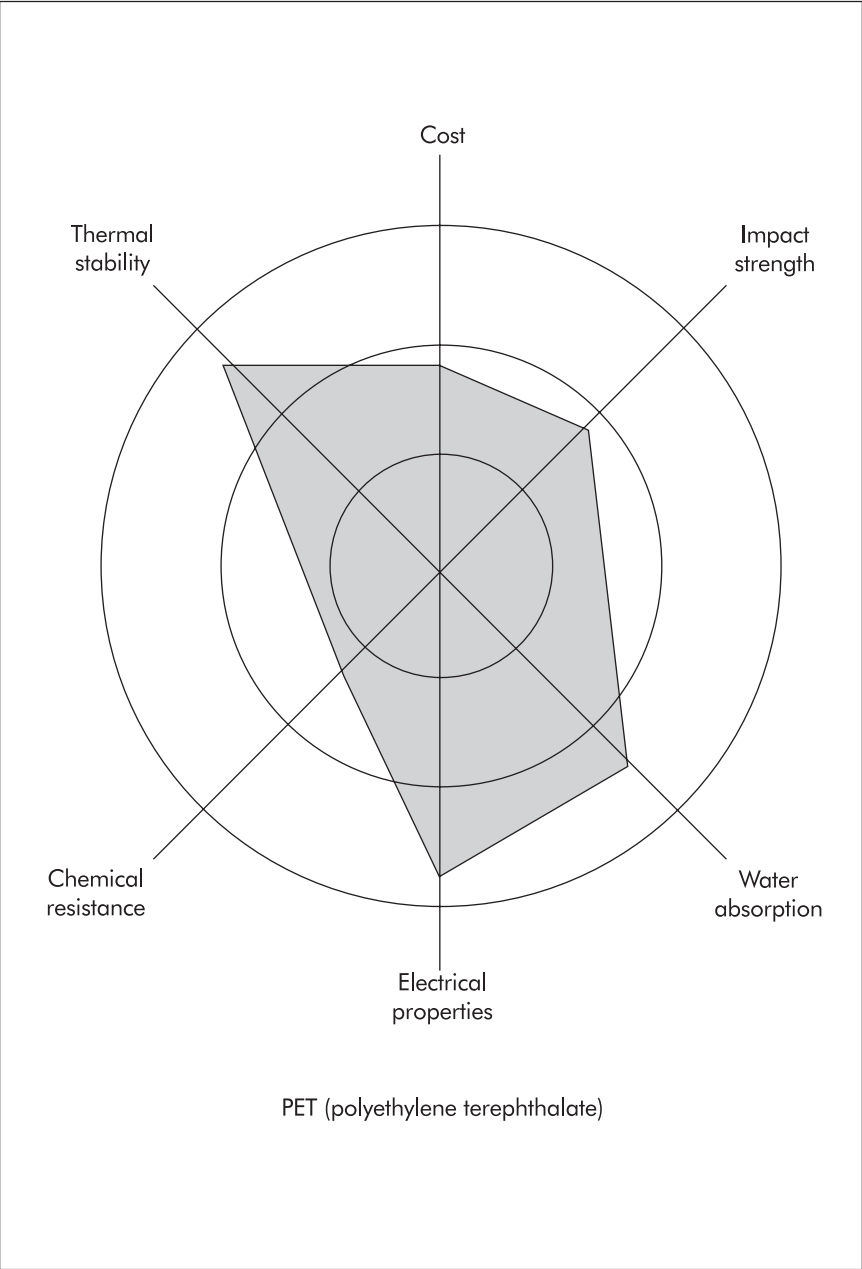


Figure 6-13. Polyethylene terephthalate (PET), a crystalline engineering grade thermoplastic, incorporates stiffness, heat tolerance, and good electrical properties. The most common grades are 30% glass-reinforced.

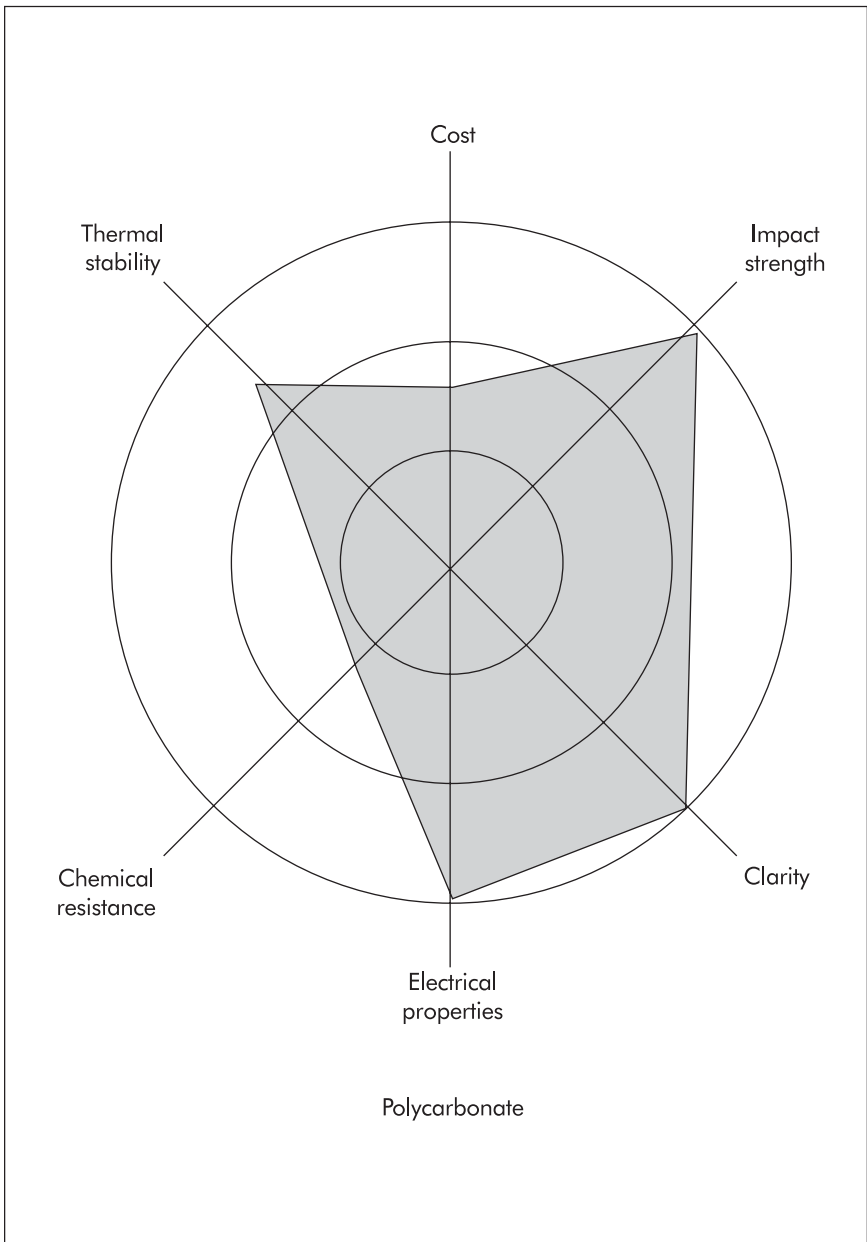


Figure 6-14. An amorphous thermoplastic, polycarbonate (PC) is a high-performance engineering material whose strongest suits are high impact strength, clarity, dimensional stability, and heat resistance. In its unfilled state it is limited by poor chemical resistance, surface hardness, and weatherability.

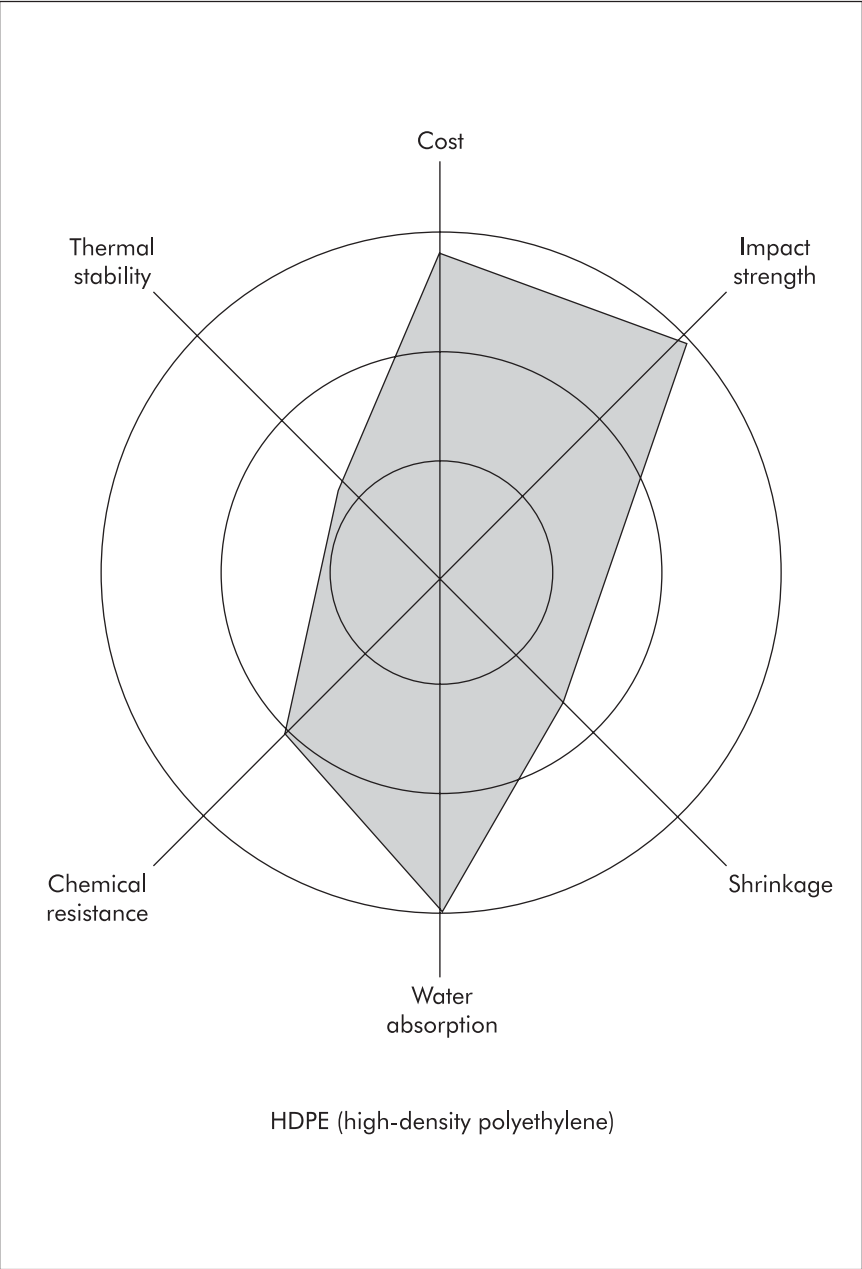


Figure 6-15. High-density polyethylene (HDPE), with a good balance of chemical resistance, low-temperature impact strength, light weight, low cost, and processibility, is a crystalline thermoplastic.

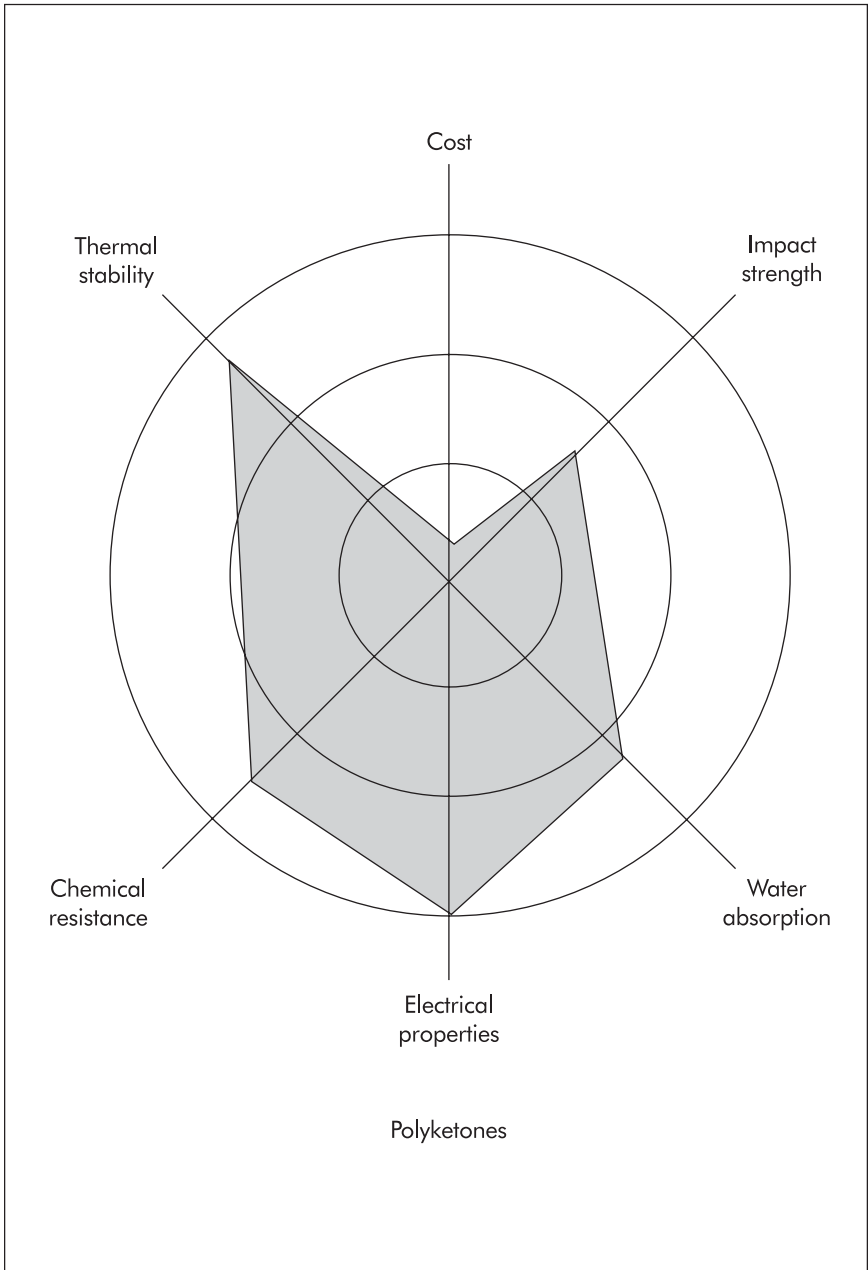


Figure 6-16. Polyketones have excellent overall property values but are extremely expensive. They are semicrystalline engineering thermoplastics with principal strengths of good thermal, mechanical, chemical, and electrical properties.

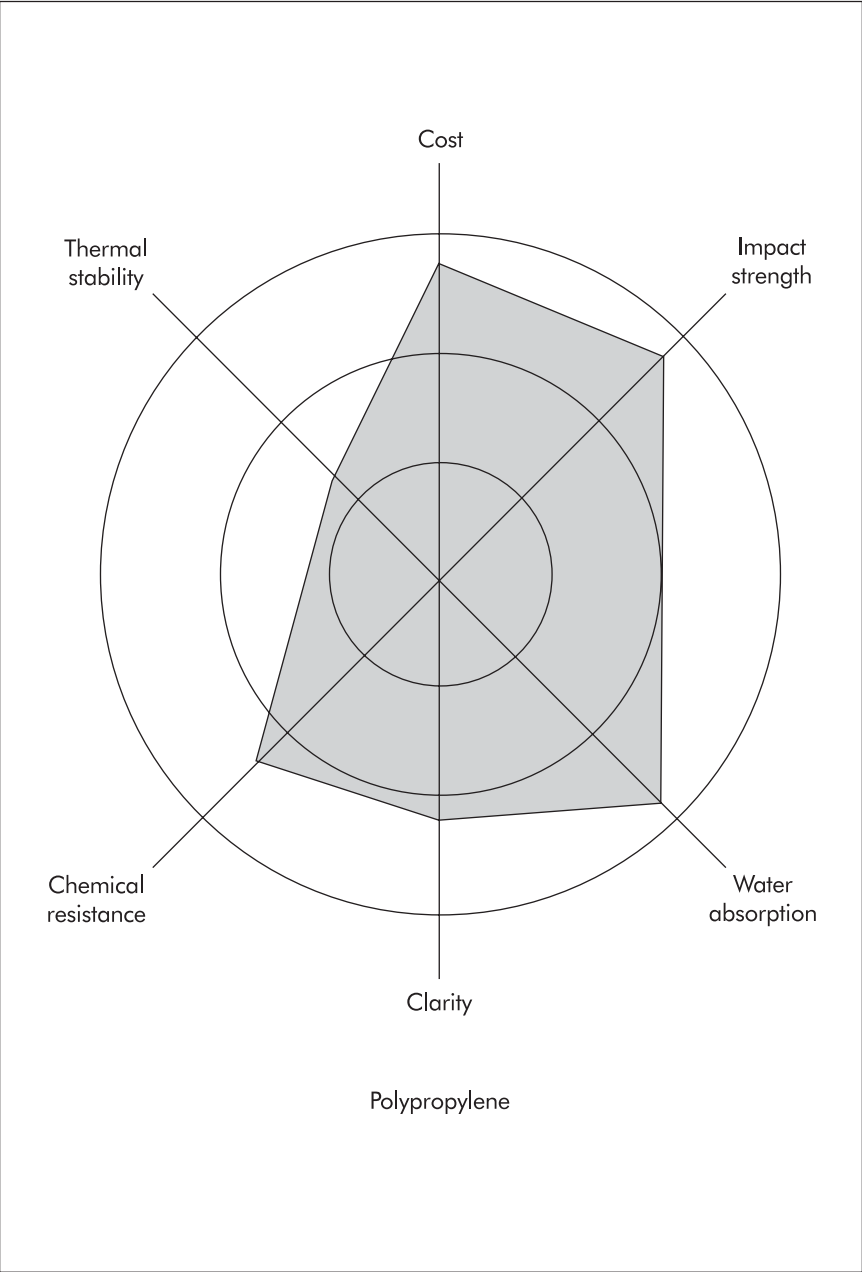


Figure 6-17. Polypropylene (PP) is a crystalline thermoplastic with a good balance of chemical resistance, low-temperature impact strength, light weight, low cost, and excellent processibility.

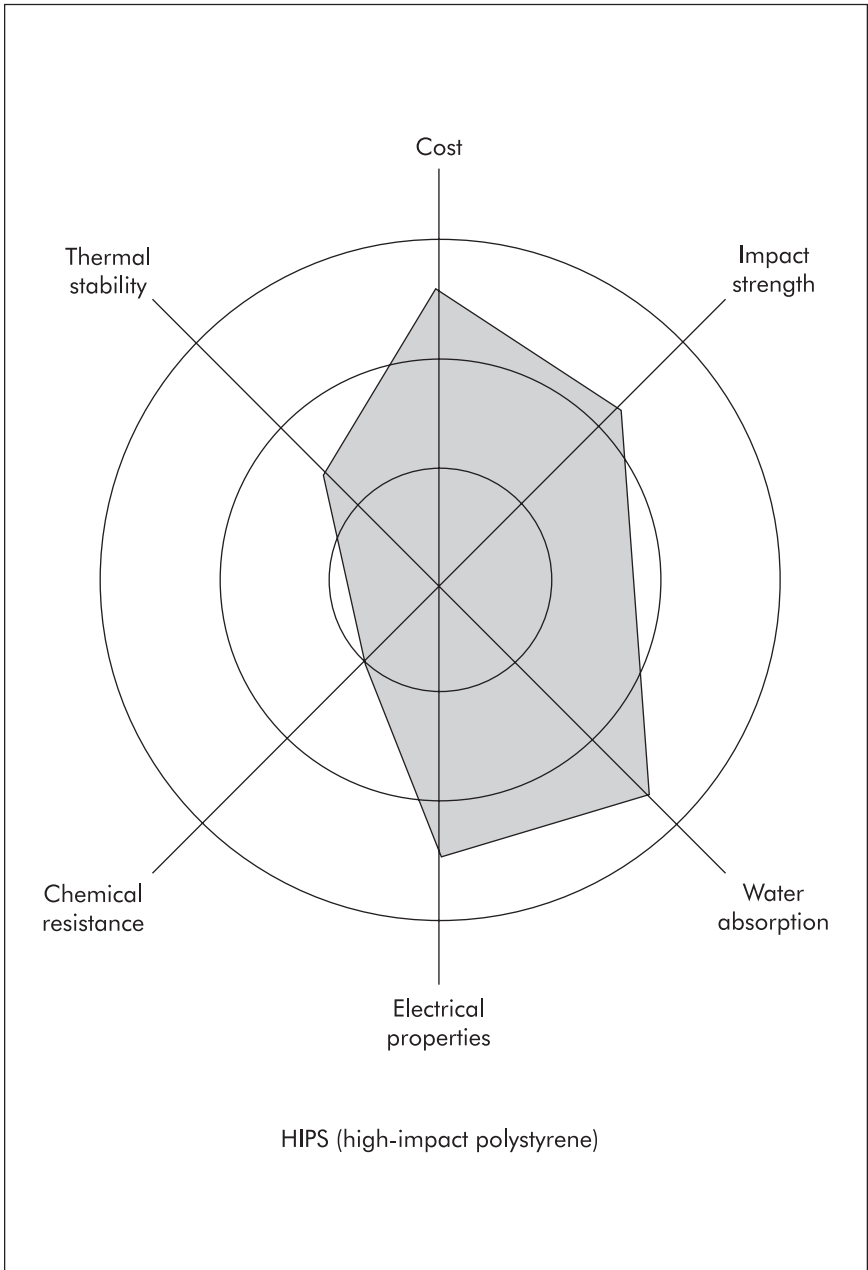


Figure 6-18. High-impact polystyrene is a rigid amorphous thermoplastic with attributes of high impact strength, dimensional stability, rigidity, processing ease, and cost effectiveness.

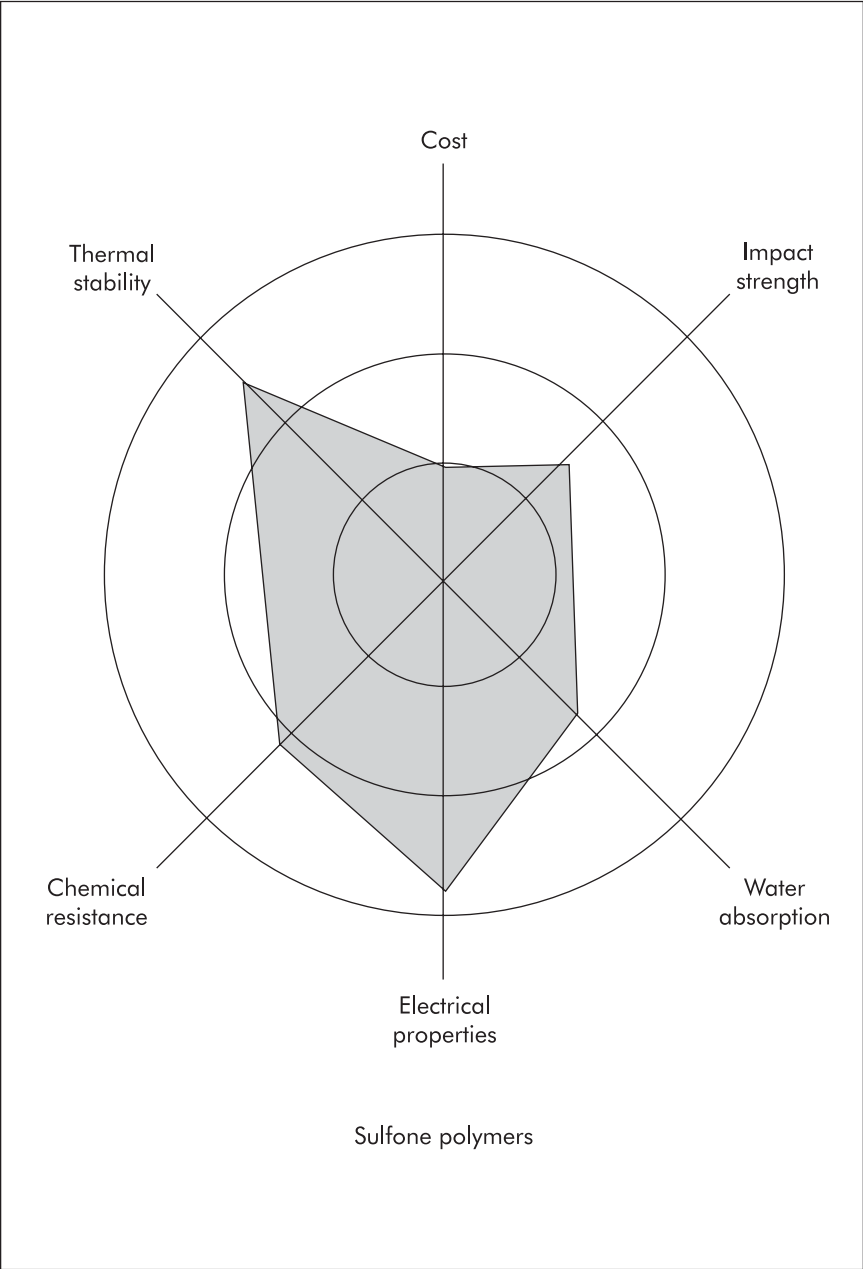


Figure 6-19. Sulfone polymers are a family of amorphous thermoplastics whose main features are thermal stability, chemical resistance, stiffness, and good electrical properties. Several of the sulfone polymers are transparent.

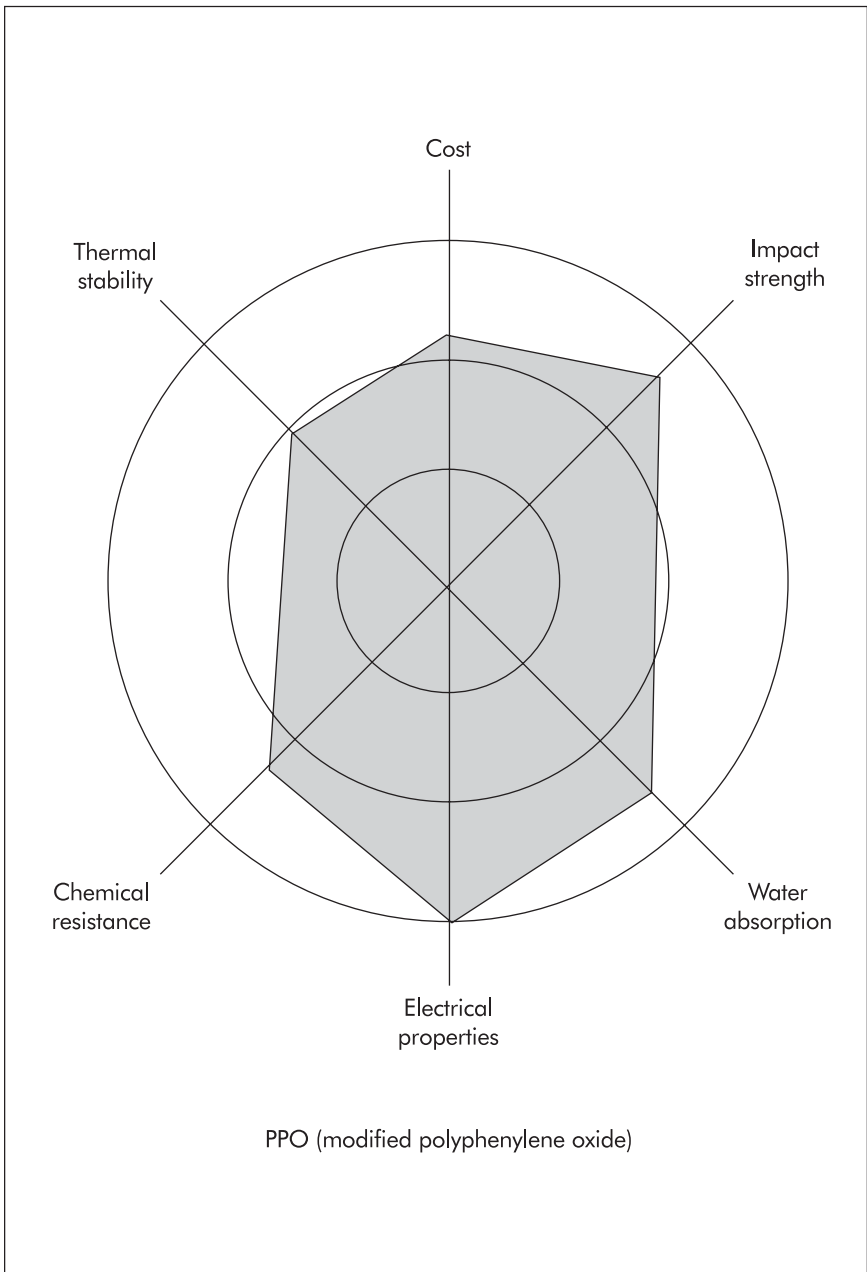


Figure 6-20. Polyphenylene oxide has good dimensional stability and electrical properties, as well as negligible water absorption. An amorphous thermoplastic, it rates high on all these criteria at either low or elevated temperatures.

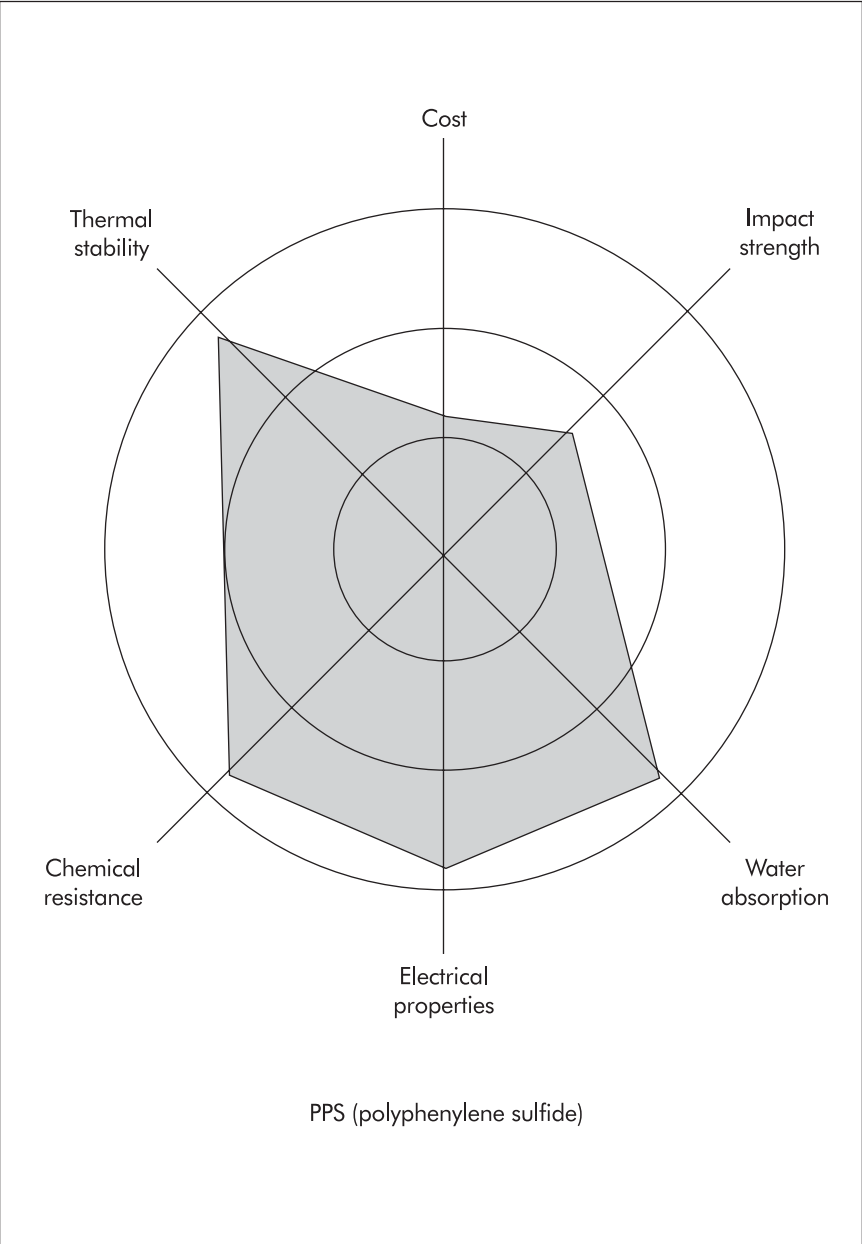


Figure 6-21. Polyphenylene sulfide is an excellent choice for products exposed to high heat or chemical environments. It is a semicrystalline engineering thermoplastic with high temperature stability, excellent chemical resistance, and good electrical properties.

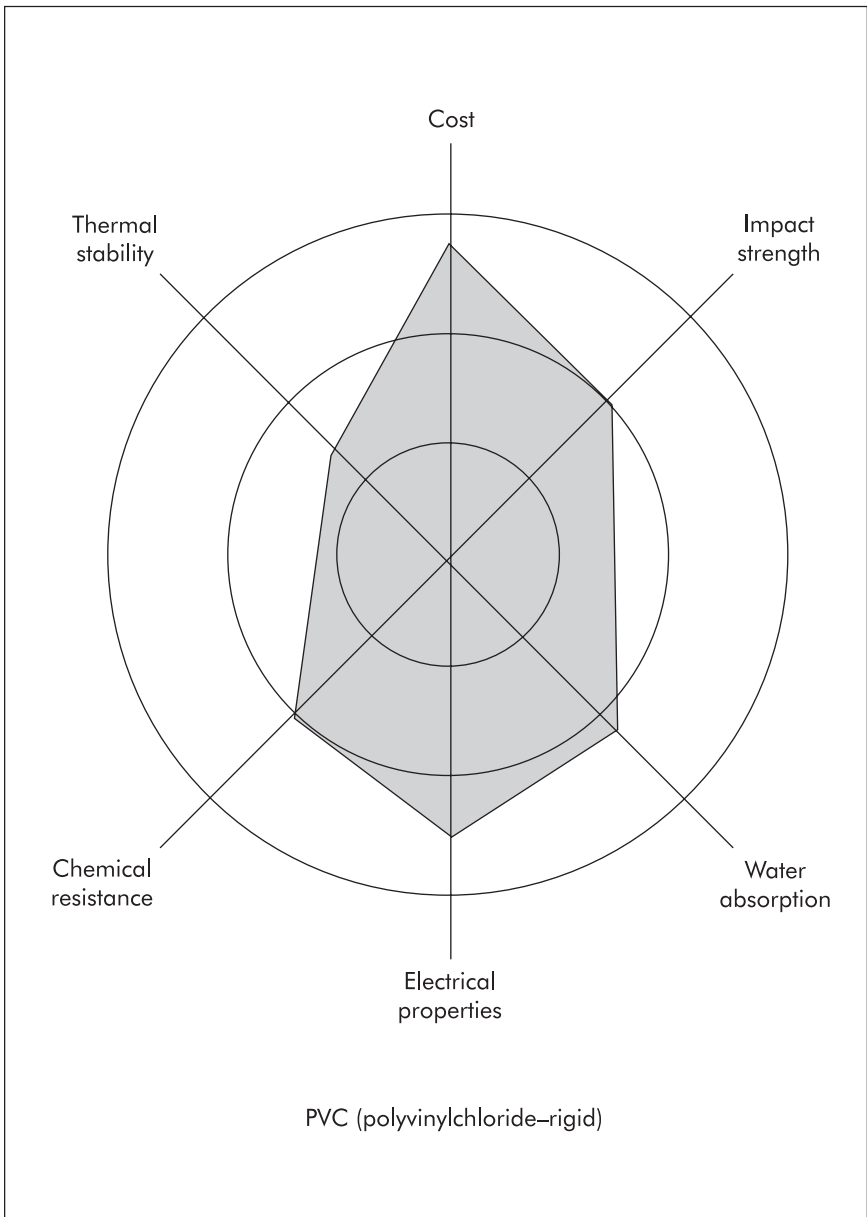


Figure 6-22. Rigid polyvinylchloride is an amorphous thermoplastic particularly suited to applications requiring weathering resistance, inherent flame retardancy, high gloss, abrasion resistance, and low cost. PVC is always compounded to some degree before processing, resulting in a wide variety of application-tailored properties.

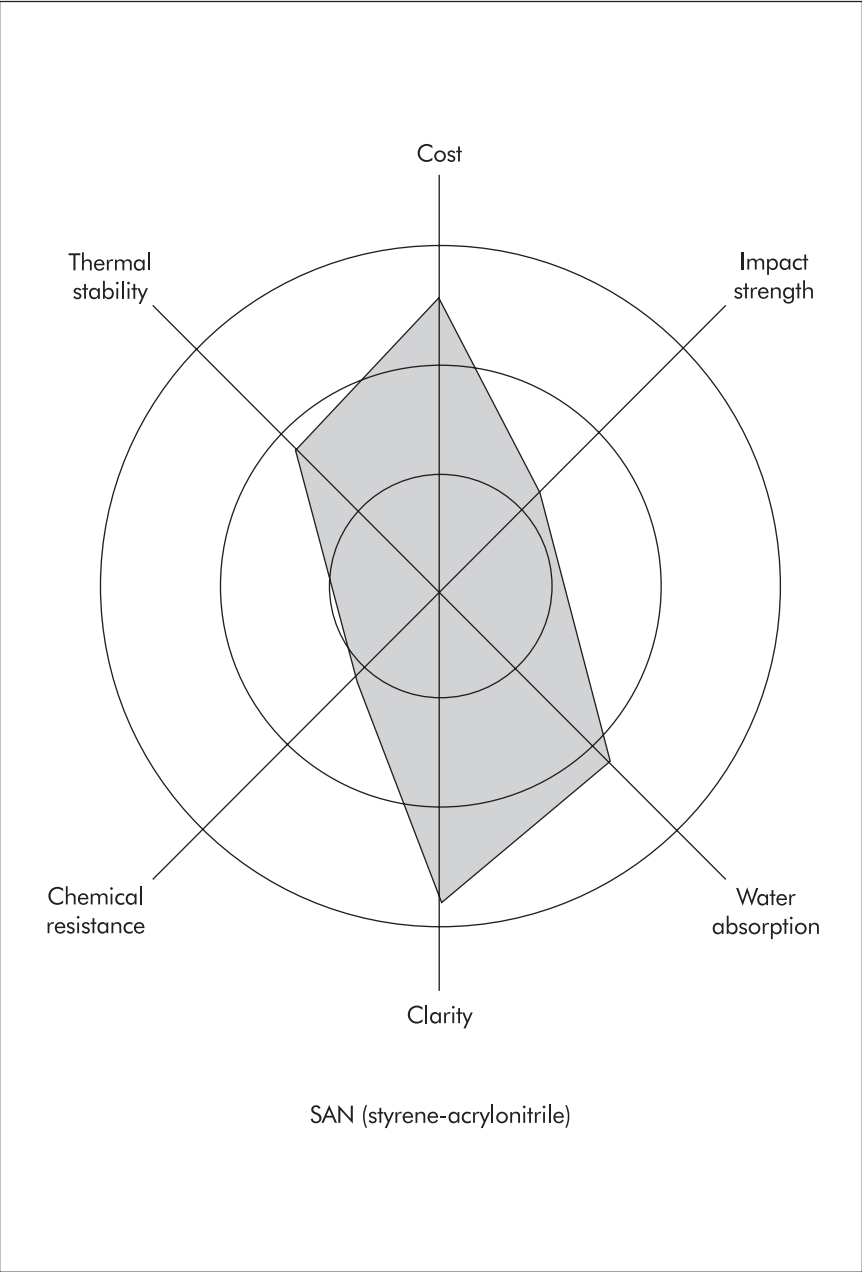


Figure 6-23. Styrene-acrylonitrile (SAN) copolymers are engineering thermoplastics whose properties include clarity, rigidity, high gloss, and low cost versus many engineering plastics.

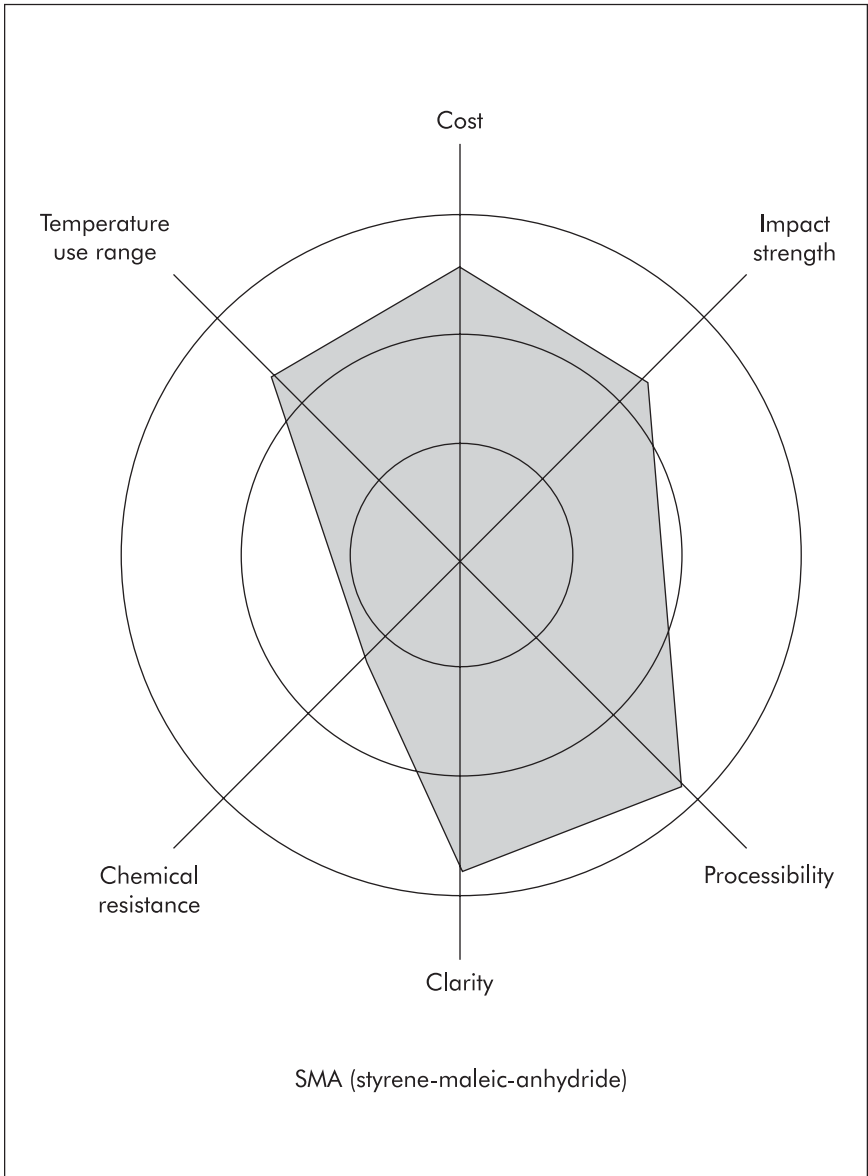


Figure 6-24. Styrene-maleic-anhydride resins provide higher heat resistance than ABS and other styrenics, while maintaining a good balance of stiffness and impact strength. Unmodified SMA copolymers are transparent and colorless and meet FDA requirements for use in medical devices as well as in food and drug packaging. Most commercial grades are rubber-modified for improved impact strength which makes them opaque.

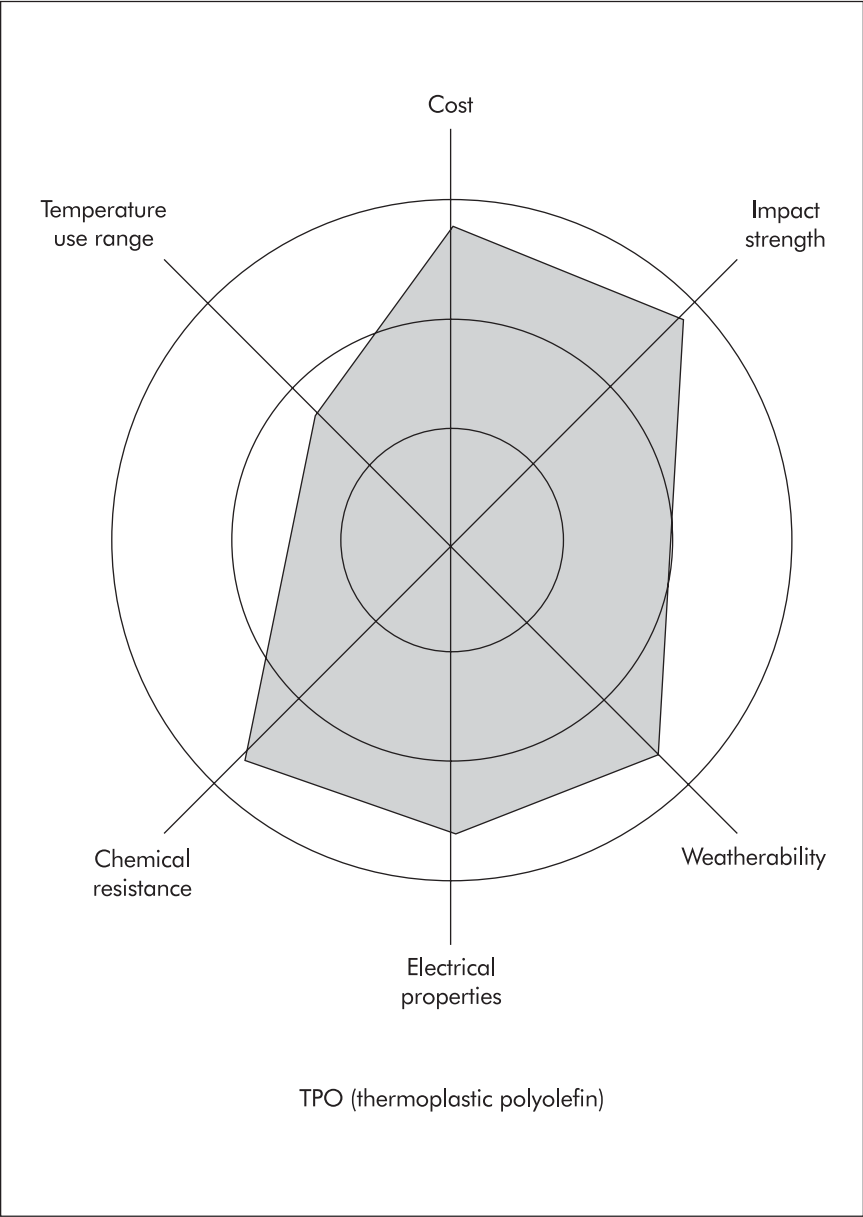


Figure 6-25. Thermoplastic polyolefins are elastomer-modified compounds, most commonly blends of polypropylene with olefin-based rubbers. While most TPOs are mechanical blends, several companies have developed methods that allow direct production in the polymer reactor. Depending on the blend ratios, TPOs can run the range from hard and rigid to soft and flexible.

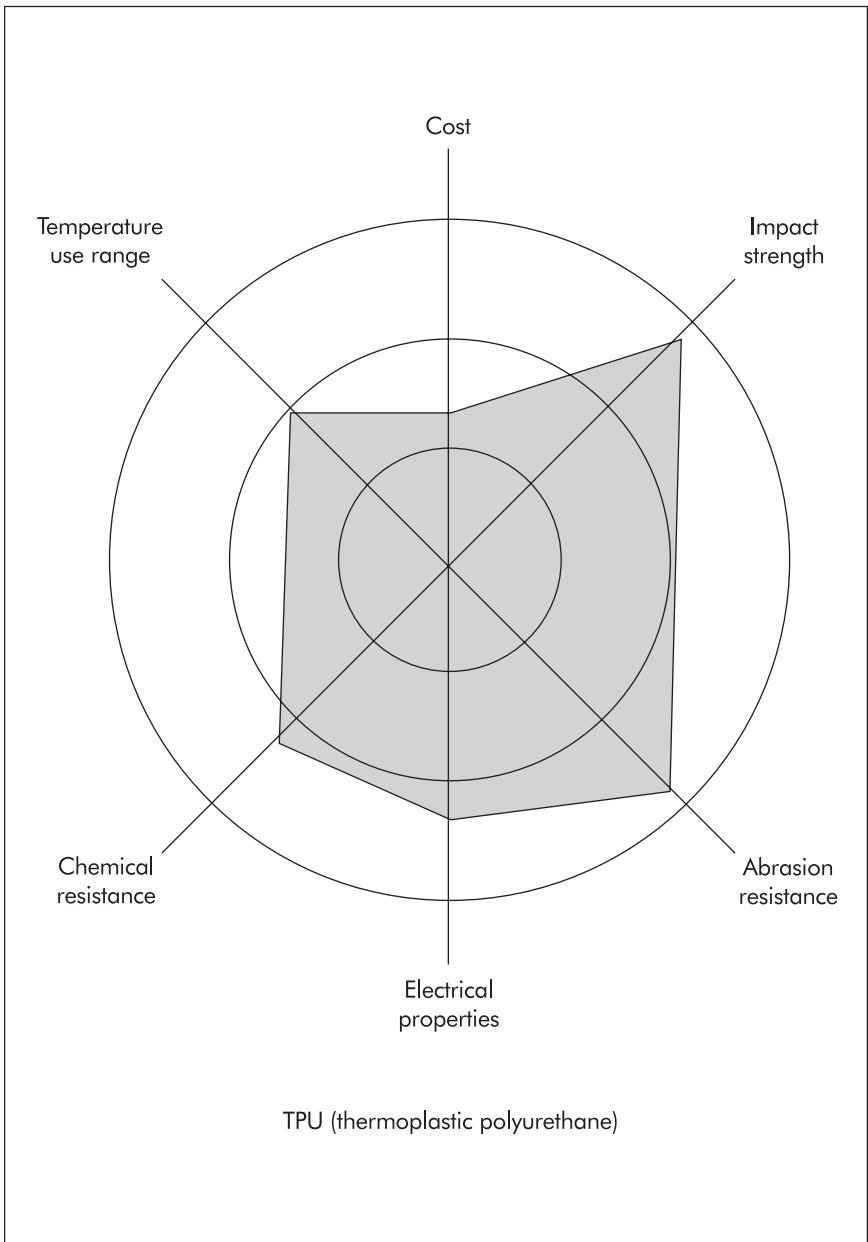


Figure 6-26. Thermoplastic polyurethane elastomers are high-performance materials whose principal features are toughness, abrasion resistance, and flexibility. They are characterized by excellent adhesion to a variety of substrates and considered at the high end of the family of thermoplastic elastomeric materials.

Determining Injection Molding Costs

7

IS IT WORTH IT?

At some point in the product development and manufacturing process it becomes necessary to establish the estimated or actual costs involved to produce a specific product. This is required for a variety of reasons, but mainly to determine if making the part is worth the cost and trouble of producing it. Accounting systems are available to determine the amount of profit expected, the competitive forces influencing the sales price, and how many pieces must be produced and sold to maintain a viable profit margin. But all of this depends on the initial cost of producing the product, so first we must calculate the actual injection molding cost to make these final determinations.

A second value of knowing how to determine injection molding costs is to be able to play “what-if” games, by changing materials, increasing or decreasing mold cavitation, adjusting volumes, or running molds in various presses. Changing one or more of these variables allows us to see in advance what cost differences exist and further allows us to alter original molding criteria to better fit our production volume models.

Finally, understanding injection molding costs provides the capability to check vendor costs to make sure they are within reason. With proper knowledge of the product, materials, and molds, assumptions can be made as to the actual manufacturing cost for producing a specific product. This can be compared to the cost of purchasing the product from a vendor and, if the difference is more than 25% or so, we can at least challenge the vendor’s cost with some feeling of expertise.

WHAT INFORMATION IS NEEDED?

Injection molding is considered a “primary” manufacturing process. For that reason, we will not address the details involved in computing secondary operations and packaging costs. We will look solely at injection molding costs. However, for convenience sake, at the end of the chapter there is a short discussion of the items that should be considered for calculating those secondary operation and packaging costs.

We first outline those items needed for calculating actual injection molding manufacturing costs. Then we discuss each item in detail.

- A. Material costs
 - 1. Raw material
 - 2. Recycled material
 - 3. Scrap allowance
 - 4. Estimated regrind buildup
- B. Labor charges (if not included in standard machine rate)
 - 1. Direct and indirect labor
 - 2. Straight time versus overtime
- C. Machine rate (hourly)
 - 1. Setup charges
 - 2. Scrap allowance and downtime
 - 3. Number of cavities in mold
 - 4. Minimum number of cavities allowed
 - 5. Cycle time per shot, in seconds
 - 6. Variable overhead (utilities, maintenance, quality control, etc.)
 - 7. Fixed overhead (rent, depreciation, salaries, etc.)
- D. Tooling charges (if amortized over product volumes)
 - 1. Initial mold costs
 - 2. Maintenance costs
 - 3. Volume for amortization calculations

Material Costs

The first item in the cost equation is material cost, an item that has options. Can we use regrind? If so, what percentage? If regrind is a viable option, how much regrind will our process generate on its own? To find this out we must estimate the volume of plastic needed to mold our parts and compare that to the volume of plastic needed to fill our runner system, assuming we are using a standard runner system. If a hot (or insulated) runner system is being used, we do not have to take into account the material needed to fill the runner. For illustration let's look at the hypothetical situation in Figure 7-1.

In this example, we will be filling a four-cavity mold, each cavity producing the same product. The standard runner system has been designed to normal specifications and the plastic material is polycarbonate. The part has a nominal wall thickness of 0.075 in. (1.90 mm).

The method for determining material costs uses the following, three-step formula:

- 1. Determine volume (in.^3 or cm^3) of part.
- 2. Determine weight per in.^3 (cm^3).
- 3. Determine cost per total in.^3 (cm^3) of part ($\text{cost/lb} \times \text{lb/in.}^3 \times \text{in.}^3$ [$\text{cost/kg} \times \text{kg/cm}^3 \times \text{cm}^3$]).

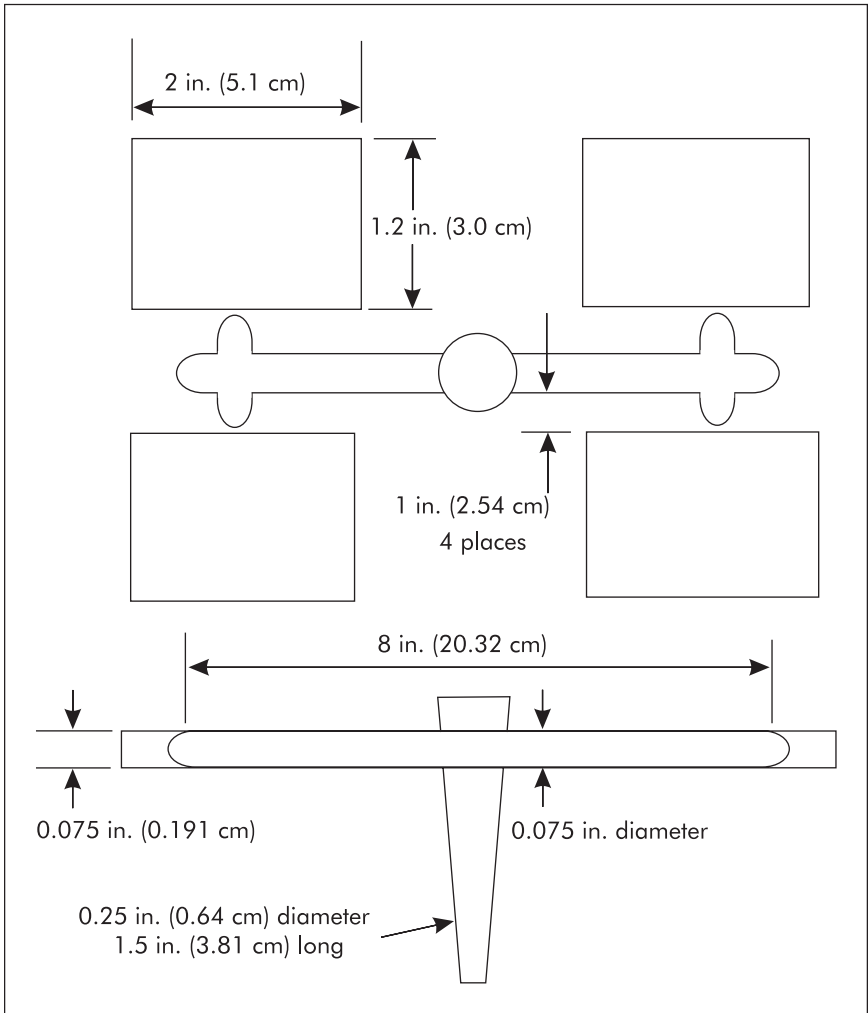


Figure 7-1. Determining volume of parts and runner system.

First, we estimate the volume of the runner system. To do this, we take the cross-sectional area and multiply it by the total length of the runner. The cross-sectional area is calculated by taking the radius of the 0.075 in.-diameter runner and multiplying it by itself (r^2), then multiplying that figure by the constant, π (3.1416). Thus, $0.075 \div 2 = 0.0375 \times 0.0375 = 0.0014 \times 3.1416$. So, the area of the runner cross-section is 0.0044 in.² (0.0284 cm²). Now, we multiply that by the length of the runner (8.0 + 1 + 1 + 1 + 1 = 12 in.) to get 0.0528 in.³ (0.8652 cm³), which is the volume of plastic needed to fill the runner. Now, we calculate the

sprue volume in the same way, but because of the taper, we can divide the answer in half. In our example, the main sprue diameter is 0.25 in. (0.64 cm), and the length is 1.50 in. (3.81 cm). Performing the same type of calculation as above, and dividing in half, we find the sprue volume to be 0.0368 in.³ (0.6033 cm³). Adding this to the runner volume we arrive at a grand total of 0.0896 in.³ (1.468 cm³) for the total runner system.

Next, we estimate the plastic needed to fill the parts. We take one part and multiply our answer by four (because it's a four-cavity mold) to get the total material required. In the case of the parts, we calculate volume by multiplying the 2.0-in. (5.1 cm) length by the 1.2-in. (3.0 cm) width to get 2.4 in.² (15.5 cm²). Multiplying that number by the wall thickness of 0.075 in. (0.191 cm), we get 0.18 in.³ (2.95 cm³) per part. Multiplying that by four (four cavities) we get a total of 0.72 in.³ (11.80 cm³) for all four parts.

So for the runner we have 0.0896 in.³ (1.468 cm³), and for the parts we have 0.72 in.³ (11.80 cm³). Our total requirement for plastic material for a single cycle is found by adding these together to get 0.8096 in.³ (13.268 cm³). To determine how much weight of plastic will be needed, we multiply that number (0.8096) by 0.0361 to get 0.0292 lb (0.0132 kg). This number is multiplied by the specific gravity of polycarbonate (or whatever material we are using) which is approximately 1.2. Multiplying 0.0292 times 1.2 we get a weight of 0.03504 lb (0.01590 kg) of plastic required for each production cycle we run. Now we take that total and multiply it by the cost per pound unit (or kg) for polycarbonate (approximately \$4.00 for purposes of our exercise) and find the cost per cycle to be \$0.140. Each of the parts (4) would then require \$0.035 worth of plastic to produce.

Use of Regrind

In most cases, the use of regrind is acceptable at specific levels and will reduce the overall cost of manufacturing the product. Purchased regrind (of high quality) can usually be procured at a per-pound cost of approximately 50% of the virgin per-pound cost. While the specific gravity may be slightly different, the total material cost of regrind can be determined using the same formulas as above.

It may be possible to use 100% regrind in a specific product, because regrind that has not been abused will retain as much as 90% of the properties of virgin. Testing will help determine the acceptable level of regrind, but the normal ratio is approximately 15% regrind and 85% virgin material.

Impact of Runner versus Shot Size

When calculating material costs it is important to understand how much regrind is being generated by the products being molded. We can then assess the use of that regrind in possibly reducing our material costs.

The normal accepted level of regrind use is 15%. If our runner system is up to 15% of the total shot size, we can use our own generated regrind. If it is more than 15% we may have to store the amount over 15% and use it elsewhere or sell it to other molders or material brokers.

In the case just discussed, our total shot size contained 0.8096 in.³ (13.268 cm³), while our runner had a total volume of 0.0896 in.³ (1.468 cm³). Taking 15% of 0.8096 shows us that we *could* use up to 0.121 in.³ (1.983 cm³) of regrind per shot. Our actual regrind volume of 0.0896 in.³ is much less than the allowed 0.121, so we can use all the regrind generated by this runner system. All we have to do is mix it back into the virgin material that we will be using. The runner system is actually molded at no cost and does not have to be used in determining the total manufacturing costs for this product. You can see by this exercise that, if we make sure our runner system is always less than 15% of the total shot size, it can be molded at no cost.

Labor Costs

Usually the cost of labor is included in the basic machine rate, but if this is *not* the case, it can be easily calculated. To do so requires that a basic hourly labor rate be established, and then added to average burden, benefit, and overhead values, which can add 100% to the hourly rate. For instance, if the average pay for a machine operator is \$7 per hour, the overhead, burden, and benefit values will probably also be worth \$7. This gives a total labor charge of approximately \$14 per hour. There are also indirect labor charges to be considered. These normally account for the supervisor, material handler, and other manufacturing personnel not directly involved with producing the product, but indirectly responsible for making sure it gets produced. The indirect labor charges are spread over the entire molding operation and only a percentage of them is added to each specific product. The total labor charge, divided by how many pieces are molded in an hour, will give a total amount to be added to the cost of each piece, for labor. And we must factor in any overtime planned to meet production schedules. This overtime rate also must be factored by the burden, benefit, and overhead values mentioned.

Machine Costs

There are a few things that need to be addressed before we can determine the actual machine costs for injection molding a product. First, the size of machine must be determined; second, the geographical location of the machine; and third, we must establish whether or not the labor cost of an operator is to be included in the machine rate. In addition, we must calculate a standard setup charge for placing the mold in the machine, getting it running for production, and removing it after the run is completed. Finally, we must decide which variable and

fixed operating costs must go into the machine rate. We will start with estimating the size of machine required.

Determining Machine Size

We determine the required machine size by establishing two things: how much clamp tonnage is necessary and how big an injection unit is needed. Both of these items are discussed in detail in Volume I of this series, so we will simply review them here.

We determine required clamp tonnage by calculating the projected area of the parts and runner being molded. Again using the example in Figure 7-1, we multiply the length of a part (2.0 in. [5.1 cm]) times the width of the part (1.2 in. [3.0 cm]) to get a projected area of 2.4 in.² (15.5 cm²). We have four cavities so we must add up the area of all four parts. This gives us (2.4×4) 9.6 in.² (61.94 cm²) of projected area for the parts. Now we add the area for the runner. The width of the runner is its diameter, 0.075 in. (0.191 cm), and the length is 12 in. (30.5 cm). Multiplying these numbers gives us a total of 0.9 in.² (5.8 cm²). Now we add the area of the runner to the area of the parts and arrive at a total of 10.5 in.² (67.7 cm²) projected area. At this point we must multiply this total projected area (10.5) by a factor representing a number of tons of clamp force. A common rule of thumb is to use 5 (5.4 metric) tons as the factor, but it can be from 2 to 8 (1.81 to 7.26 metric) tons, depending on how easily the plastic flows when it is injected. Easy-flow materials require less injection pressure, thus lower clamp pressure to hold the mold closed against the injection pressure. The flowability of a plastic is determined by its melt index rating and is covered in detail in Volume I of this series. Our material of choice, polycarbonate (which is fairly difficult to flow), has a factor of 5, so we will multiply our projected area (10.5) by a factor of 5 (4.54 metric) tons. This equals 52.5 (47.7 metric) tons, and represents how much total clamp force will be required to hold the mold closed during the injection process. We should add a safety factor of 10% to this number and find a machine that will give us at least 57.75 (52.39 metric) tons of clamp. We would probably place this mold in a 60 (55 metric)-ton machine. If we do not have the exact size machine available we can place it in a larger one, but *never* in a machine that exceeds 10 tons per square inch of projected area, as that is enough to collapse the steel mold and press the mold into the machine platens, damaging the mold and machine. In our case that would be a machine with 10 tons times our projected area of 10.5 in.² (67.7 cm²), for a total of 105 (95 metric) tons. Our machine range then is 60 to 105 tons.

Next, we want to determine the injection unit requirements. This, too, is calculated using the projected area of the runner and parts. The number above, 10.5 in.², is the one we will use.

All plastic materials have a specific flow rate. Some flow easily, which means they do not require high injection pressures, and some are more difficult to push, requiring higher injection pressures. An average value for injection pressure is 10,000 psi (68,950 kPa). Because “psi” stands for pounds per square inch, we can see that injection pressure is rated at how much force is needed over a certain area, notably the projected area we calculated for the parts and runner. So we must have a machine capable of producing 10,000 psi injection pressure. Most machines are designed to provide at least 20,000 psi (137,900 kPa), so this should not be a problem.

We must now determine the exact amount of material needed to inject during one cycle. We have already calculated that to be 0.8096 in.³ (13.268 cm³), or 0.03504 lb (0.01590 kg) of plastic required for each production cycle we run. Machines are rated as to how many ounces (grams) of material they are capable of injecting at one time, so we must convert our number to ounces by multiplying it times 16, which gives us 0.560 oz (15.88 g), or just over half an ounce. An ideal situation allows us to inject half the barrel capacity every shot, so we would want to place our mold in a machine with a 1.12-oz (31.75-g) barrel. Of course we will look for a standard size which would be 1 oz (28.35 g). There is a standard rule of thumb that states that we should run a mold in a machine so that we use anywhere from 20 to 80% of the barrel capacity per shot. In our case that would allow us to use any machine with a barrel size from 5/8 to 2.5-oz (17.72 to 70.9-g) capacity (1/2 oz is 80% of a 5/8-oz machine, and 20% of a 2.5-oz machine).

So, now we know that our four-cavity mold should be run in any machine with clamp tonnage from 60 to 105 (65 to 95 metric) tons, and with a barrel capacity of 5/8 oz to 2.5 oz of plastic material. Now let’s consider the overhead costs.

Under the classification of *variable* overhead costs are items such as utilities (electricity and water), maintenance expenses (machine), and quality control systems. These are considered variable because they tend to change from product to product and season to season. Actual expenses can be determined by reviewing operating cost records; specific costs can then be estimated for similar conditions and specific products. But typically, an average is calculated, and this average is added to all products as part of the standard machine rate factor. Utility costs average approximately 40% of the total operating factor, with maintenance costs at approximately 10%, and quality control systems at approximately 8%.

Fixed overhead costs include items such as rent, machine depreciation, some of the management salaries (those closely related to the manufacturing process), and other costs that do not tend to fluctuate over time and are considered

necessary for the production of a specific product. In total they typically average 25% of the total manufacturing cost. Again, these are usually spread out over all of the pieces of manufacturing equipment rather than trying to determine how much is expended for each piece of equipment.

Determining Machine Location

Utility rates and operating costs differ in different parts of the U.S. In the Northeast and far West, they are the highest, and in the South Central and Southeast they are the lowest. This means that the cost of doing business changes with changes in geographic location. The machine hour rate (MHR) can be defined as the hourly costs involved for the operation of a machine and includes such items as overhead, management salaries, plant maintenance, etc. The MHR will reflect these costs dependent on where the machine is geographically located. Manufacturing costs must be related to the location of the actual machine. Table VII-1 shows selected rates.

Table VII-1. Average Machine Hour Rates

Machine Size	Cost per Hour
0 to 100 ton	= \$25 to \$30
101 to 200 ton	= \$30 to \$35
201 to 300 ton	= \$35 to \$45
301 to 500 ton	= \$45 to \$55
501 to 750 ton	= \$55 to \$75
751 to 1000 ton	= \$75 to \$100

Rates in Table VII-1 are average costs and normally include one operator. For far West and Northeast locations add 25 to 50%, depending on local utility rates and wages. For South Central and Southeastern locations, deduct 15 to 25%, depending on the same factors.

As can be assumed from this information it makes good sense to place a mold in the smallest machine possible, to keep the manufacturing costs low.

Now that we have critical machine hour rates we can determine molding costs for our four-cavity mold.

Calculating Molding Process Costs

To arrive at process costs, we need to estimate an overall cycle for our production run. The cycle is based on many things (addressed in Volume I), but what we are interested in here is the total, or gate-to-gate cycle. The main item of importance in determining cycle times is the time it takes to cool the plastic,

which is dependent on wall thickness. Overall cycle times for a sampling of various wall thicknesses are:

0.060-in. (0.152-cm) wall thickness	=	18 seconds
0.075-in. (0.191-cm) wall thickness	=	22 seconds
0.100-in. (0.254-cm) wall thickness	=	28 seconds
0.125-in. (0.318-cm) wall thickness	=	36 seconds

These cycle times are *average*. If the mold is complicated (such as an unscrewing mold), or large, the cycles may be much longer, owing to extended open and close portions of the cycle.

Now that we have a guide for estimating cycles, we can estimate the number of pieces produced per hour. This is done by dividing the number of seconds in an hour (3600) by the gate-to-gate cycle for our product, and multiplying that by the number of cavities being molded at one time. The formula is:

$$(3600 \div \text{cycle}) \times \text{number of cavities}$$

Still working with our earlier case, we determine from the cycle time chart that an average cycle for a part with 0.075-in. (0.191-cm) wall thickness is 22 seconds. We divide 3600 by 22 seconds to get 163.63. This we multiply by the number of cavities we have (four) to get a total of 654.52 pieces produced in an hour's time. If we run our mold in a 100 (90.72 metric)-ton machine, the hourly rate for that size machine is \$30 (from Table VII-1). So we know that it would cost \$30 to produce 654.52 molded parts. Dividing the \$30 by the number of parts (654.52) gives us a cost of \$0.046 per piece. At this time we should add a scrap factor, usually 10%, to the value of the parts. For us that would be \$0.005. Adding that to our cost we arrive at a molding cost of \$0.051 each.

Should Operator Cost be Included?

Opinions differ on including operator cost. Each case should be studied and assessed on its own merits. If a molding facility has a set number of operators (such as might be the case in a captive shop) and does not wish to fluctuate that number, then it would be wise to include an operator cost in every quote. On the other hand, if the company has seasonal requirements and uses robots for most of the molding machines, then the cost of an operator should not be included. This may give them an edge in obtaining new business.

If a facility is not labor-intensive, it must use automated equipment or sophisticated tooling to achieve the necessary production requirements. By doing so, extra costs are involved that must be included in determining the manufacturing cost of any product.

So in either case, (using an operator or not) there are costs involved, which usually cancel each other out and do not greatly affect the actual machine hour rate.

Estimating Setup Charges

A setup charge is a one-time fee assessed for each time a mold is placed in a press for a production run. Normally it is derived from establishing the amount of time required to make the setup and multiplying that times the hourly rate for that machine. Thus, a 2-hour setup on a 300 (272 metric)-ton machine would be approximately \$90 (2 hours \times \$45). The normal time to complete an average setup is from 2 to 3 hours. There are many quick-change devices available today designed to reduce that time to a matter of 15 minutes or so, but the cost of utilizing that technology adds considerably to the investment required to achieve this. However, the investment may be worthwhile if it means minimizing operation costs to improve competitiveness.

For companies that have high volumes of production, or leave a specific mold in a machine for long periods of time, the cost of the setup for that mold is usually absorbed. But in cases where volumes are smaller, or molds are changed frequently (such as in most custom molding facilities) a setup charge is usually levied. This charge could run as little as \$50 or as much as \$500 or more, and is usually invoiced separately from the cost of molding the products. The decision to do so or not is entirely up to management who must take into account such things as customer loyalty and level of capacity at the time. In some cases, especially for low volume runs, the setup charge is amortized over the total number of pieces being produced. In other cases it is absorbed in overhead and considered a standard part of doing business.

Tooling Costs

Tooling costs are extremely hard to estimate because of the variety of designs and steel types available. Tools can range from a few thousand dollars to many hundreds of thousands of dollars depending on these factors, as well as the size of the part being molded and how many cavities are being molded at one time (called cavitation). Today, moldmakers are charging in the area of \$55 per manhour to build molds. The average time for building a mold is around 1,000 hours. So, an average mold will cost around \$55,000 to build, not including cost of steel (another \$5,000). While these may represent *average* costs, it is not wise to depend on them for a specific project because of the many variables already listed. Most moldmakers will be happy to give a “ballpark” figure that can be used for budgetary purposes, without the potential customer running the legal risk of asking for a formal quote but never intending to release a purchase order at that time.

Tooling costs may or may not be included in the manufacturing cost estimates. They may be paid for by a special fund dedicated to total tooling costs for a specific plant site. Regardless of how the tooling is funded, there is a

definite cost associated with building molds and secondary tools for a specific product design.

Standard Practice

In most cases, tooling (mold) charges are *not* amortized over a specific volume. This is because volumes never stay fixed and are continuously changing as market influences change. To keep from having to constantly adjust manufacturing costs based on these changes, molders normally treat the cost of tooling as a separate issue. Usually the billing practice for tooling costs requires that the customer pay a third of the cost as a down payment to the moldmaker, a third upon delivery of the mold to the molder, and the final third within 30 days of tool acceptance by the customer. This practice keeps the moldmaker from being burdened with funding the initial building of the mold and risking loss of capital, which can run into hundreds of thousands of dollars. The toolmaker (moldmaker) is responsible for building a mold that is capable of producing a specific product to print. There are many variations to this procedure, however, and it makes good sense to determine ahead of time just what the moldmaker is responsible for and what constitutes an “acceptable” mold. For instance, if the customer dictates the amount of shrinkage to be used in calculating mold dimensions, the moldmaker cannot be held responsible if the molded part dimensions are incorrect.

Amortizing Tool Costs

If tooling costs must be amortized, for whatever reason, it is usually done so over the first year’s production run, which has been negotiated and *guaranteed* by the customer. That way, even if the parts are never produced, the molder and moldmaker are paid for their involvement. This practice results in unusually high costs for the first year’s molded parts, with each part absorbing a portion of the total tooling costs. If the run is only a few hundred pieces, each piece may become so costly it cannot be sold. This is the primary concern regarding the amortization of tool costs.

Maintenance Costs

Whether tooling costs are amortized, or paid for up front, there is a need to address everyday maintenance charges for upkeep of the mold. As a rule, this will result in an annual charge that is equal to approximately 8% of the original tool cost. For example, if a mold costs \$50,000 to build, we can assume that we will spend approximately \$4,000 every year to maintain that mold. This money is used for such things as waterline hoses, ejector pins, lubricant, damage repair, and other items required to keep the mold running in top condition. This money can be included in standard overhead charges, added to the original cost of the

mold, or billed for as the items are activated. If the volume of production is great enough, the charges can be added to the piece price (amortized) and absorbed by the molder as required.

Adding It All Up

At this point we have enough information to determine the actual costs involved to mold our product. All we need to do is add the material costs (\$0.035 per part) to the molding cost (\$0.051 each) to arrive at \$0.086 each. If there are any secondary operations to be performed, those costs would be added to this figure. *Secondary* operation costs are determined in the same manner as molding costs: by determining pieces per hour and any material costs. In addition, packaging, inspection, and freight charges may have to be added. But the primary manufacturing cost has been determined to be \$0.086. This number can also be used as a “sanity” check to determine if a vendor is charging a reasonable price for molding a product for you, the customer, or to develop a “target” cost for in-house molding operations. Remember though, that no profit has been included in this number. With a properly designed spreadsheet, the information detailed in this chapter can be manipulated to determine how the final costs are affected by such things as a change in machine size or an addition (or deletion) of cavities. Also, it may be practical to check on costs for molding in one specific geographic location versus another, if that flexibility is available. The potential cost savings may be worth the move.

Secondary Operations (Including Packaging)

In Chapter 8 we take a close look at typical secondary operations, including surface finishes. These are called secondary because they are performed after the primary operation of injection molding. Even activities such as inspection and packaging can be considered secondary operations. Sometimes these operations can be performed on parts during the following molding cycle, in which case they are virtually free. In other words, the parts from one cycle can have operations performed on them while the next cycle is producing the next set of parts. This is not always possible because of cycle time constraints, but if it can be done while the operator is standing, waiting for the cycle to end, it can be done for no additional cost by using that operator’s free time. Sometimes, molding machines are operated automatically, without an operator. In that case, it may not be economically advantageous to have an operator at the molding machine simply to perform secondary operations. It would usually be more cost and time efficient to have the operator perform secondary operations in a location designed specifically for the secondary operation.

Costs of all secondary operations should be calculated. Whether it is assembly, machining, painting, sanding, inspecting, or packing, the activity takes a

certain amount of time to perform and requires a specific amount of material and equipment to do so. In most cases, it is adequate to estimate the cost of labor to be actual operator time plus a factor of from 1.5 to 3. That factor is intended to include overhead, burden, and certain facility expenses such as restroom maintenance. The higher the level of these expenses and costs, the higher the factor used to determine labor costs. A normal factor is 1.75. If operator wages for secondary operations is \$8.00 per hour, normal factoring brings that figure to approximately \$14.00 per hour ($\8×1.75). Then we must consider the machine hour rate for the operation being performed. Like the MHR for the molding equipment, this number comes from the financial department and is based on the original cost of the equipment, the expense incurred in operating the equipment, and the amount of floor space taken by the equipment, operator, and work-in-process (WIP). For small operations like inspection, this rate can be low, in the area of \$2 to \$3 per hour. But in the case of sophisticated operations like automated deflashing or surface finishing (painting), the cost can be as much as \$100 per hour. It can be seen by these numbers that it is quite advantageous to perform secondary operations at the molding machine, if time and space allow.

Packaging is usually not considered a secondary operation, but it should be. It meets all the criteria as we defined earlier. The first item to consider is the size of the container being used. This is usually calculated by a packaging team or expert, who strives to find the best economical matches among the size of the product being packaged, the weight of the final sealed container, the method of shipment used, and the number of products required in an average shipment. The packaging process itself must consider the ideal method of placement of the product, the space utilized for performing the action, the number of products required per container, and the materials used for the activity. Again, the cost of the operator should be calculated using the hourly wage times a factor of 1.5 to 3, depending on the level and amount of overhead, burden, and facility expenses intended to be absorbed by the packaging operation.

SUMMARY

Understanding how to determine injection molding costs can be beneficial in three ways:

1. To determine if making the product will be profitable,
2. To play “what-if” games by changing materials, cavitation, volumes, or press sizes to see if additional profit can be generated, and
3. As a monitor to determine whether or not outside molding vendor costs are reasonable.

There are four basic items to be analyzed for determining molding costs: material, labor, machine rates, and tooling costs.

When calculating material costs, the volume, or *cost per cubic inch* is the factor to look for, not just the cost per pound of raw material.

If possible, it is wise to utilize regrind plastic, as the price is usually half the cost of virgin material.

The amount of regrind that can safely be utilized is usually 15% of the total by weight. However, tests have shown that 100% regrind can be just as good as 100% virgin concerning final physical properties.

Unless hot runner systems are being utilized, the amount of material that makes up the runner and sprue should never be more than 15% of the total shot size, by weight. Following this rule of thumb will ensure that there is no more than 15% regrind material available for use.

Clamp tonnage determines the size of machine needed for molding. This is calculated by determining the projected area of the cavities and multiplying that number by a factor from 2 to 8, which is the number of tons required to keep the mold closed against injection pressure.

The machine hour rate, or MHR, fluctuates because of geographical location, with the Northeast and far West being the most expensive areas in the U.S.

The total molding cycle is primarily dependent upon the cooling portion of that cycle.

Tooling costs are not normally amortized over the life of a product, but rather are paid for up front in a lump sum.

QUESTIONS

1. Why is it important to understand how to determine the cost to injection mold a product?
2. What are the four basic items needed for determining these costs?
3. Why is it important to know the cost per in.³ (cm³) rather than simply cost per pound (kg)?
4. What is the maximum percentage of regrind that can be used, and why?
5. When using a standard runner, what should be the maximum percentage of runner and sprue versus the total shot size (by weight)?
6. What are the two primary items of information required to determine which machine can be used for molding a specific product?
7. What is the normal amount of maximum injection pressure available on most molding machines today?
8. In an ideal situation, how much of the material in an injection barrel should be injected every cycle?
9. What percentage of the average total operating costs for molding is made up of utility costs?
10. What two areas of the United States are most expensive when determining injection molding costs?

8

Secondary Operations

DEFINING SECONDARY OPERATIONS

A secondary operation can be defined as any operation performed on a product after it has been molded. Such operations normally include, but are not limited to, assembly, machining, and finishing (including decorative finishes). In this chapter, we look at the fundamental methods of performing these operations.

It should be noted that secondary operations usually mean increased product cost. Most secondary operations are performed outside the normal molding cycle time and usually require additional personnel. In some cases secondary operations are performed right at the molding machine by the machine operator if time allows it to be accomplished within the normal molding cycle time.

Because of the additional cost to perform secondary operations it must be stated that *all secondary operations can be eliminated through part design and mold design, if cost and time are not a consideration.*

For instance, a part can be decorated within the mold using existing technology that actually deposits the required finish on the mold surface, and then transfers it to the product during the molding cycle. Another example would be assembly. In this case, two parts could be molded side by side and snapped together while the mold is ejecting them. Or one part can be molded, then insert-molded into the second part. In any such case, the cost of designing and building the secondary capabilities into the mold are expensive and can add 200 to 1,000% to the normal mold costs. It can also add the same percentage to the delivery time for completing the mold build. These factors must be carefully considered when determining whether or not to build the secondary operations into the mold rather than performing them external to the mold.

Another consideration is the use of robotic systems to perform secondary operations outside of the mold. This reduces the cost of the mold but increases the financial investment of performing the secondary operations and increases the amount of processing and storage space requirements. Financial analysis will aid in making the final determination.

WHEN TO CONSIDER SECONDARY OPERATIONS

There are times when secondary operations are preferred to using mold design to incorporate the secondary operation requirements.

When Volumes are Small

In the injection molding industry, annual product volume requirements of less than 25,000 pieces constitute what is called “low volume.” In some cases this number may be as high as 50,000, but in either case these are considered small numbers.

When a mold is built, it is common practice to understand that the cost of that mold must be absorbed in the selling price of the product being produced. A convention in the industry is that it is less expensive to perform secondary operations on a molded part than to increase the complexity—and cost—of the mold when the annual volumes are less than 25,000 pieces. This convention assumes that the life of the product will not exceed 3 or 4 years.

When Tooling Costs are Excessive

Moldmaking shops work under the same market demand concepts as any other manufacturing operation. This results in their prices fluctuating to what the market will bear. In some cases the cost (and delivery time) of a mold may double simply because it is requested during one of the higher pricing phases of the economy. In a case such as that, the buyer may elect to reduce the amount of “molded-in” requirements of the mold, and perform some secondary operations simply as a way of reducing the initial cost. Of course the buyer could wait for the prices to come down, but this would undoubtedly result in a loss of market share or even a totally missed opportunity due to delayed timing.

In addition, most projects are funded based on early budget-setting exercises which may dictate the total amount of money available for building a mold. The buyer will get everything he or she can get built into that mold for the money available, but it may not be enough to cover all the requirements. In that case, some of the functions or features may have to be accounted for in secondary operations.

When Time to Build the Mold Jeopardizes Marketing Schedules

Lead time for moldmaking is a common problem, becoming more common as development cycles and expected product life cycles get shorter and shorter. Usually, the greatest amount of time taken up by a single item during the development cycle of a molded product is the building of the injection molds. This can take from a few weeks for small, simple products to many months for complicated or large products. Normally, the amount of time allocated by early estimators for moldmaking is too short, and the final result is a need to make

the mold design-and-build phase much less complicated than may be desired, simply to meet marketing schedules. In that case, some of the built-in features of the mold will have to be eliminated and those features accounted for through the implementation of secondary operations.

When a Labor-heavy Environment Exists

There may be occasions when a company has an overabundance of employees on the payroll, possibly because a large contract was unexpectedly canceled or marketing forecasts were negatively affected by economic conditions. The company may have spent a large amount of money and time training these employees and, believing that they will need the same employees in a short period of time, may wish to keep them rather than lay them off and have to go through the hiring and training process all over again later. At times like these, it may be economically advantageous to utilize the retained employees to perform secondary operations, thus reducing the initial mold investment and delivery time, and maintaining a level payroll.

ASSEMBLY OPERATIONS

A variety of assembly operations exists, including snap-fitting and use of screws and bolts. The following paragraphs detail some of the more common thermal and mechanical methods of assembling that go beyond the snap-fits and screws. A section on ultrasonic welding details the variables that affect sonic welding, but these same variables generally apply to the other forms of welding as well.

Ultrasonic Welding

Ultrasonic welding is an assembly process that utilizes high-frequency mechanical vibrations (20 to 40 kHz [20,000 to 40,000 cycles per second]) transmitted through thermoplastic parts (thermoset materials cannot be ultrasonically welded in the traditional sense). These vibrations generate friction between the plastic parts being assembled, and this friction causes heat. The heat generated causes the mating plastic surfaces to melt slightly and fuse together, resulting in a welded product. The ultrasonic welding process can be used for staking, surface (vibration) welding, spot welding, and installing metal inserts. A typical ultrasonic welding machine is shown in Figure 8-1.

Both amorphous and crystalline materials can be ultrasonically welded, but crystalline materials require greater amounts of energy and are much more sensitive to joint design, horn design, and fixturing. Basically, the higher the melt temperature, the more ultrasonic energy required for welding. The major factors affecting weldability include polymer structure (amorphous versus crystalline), melt temperature, melt index (flow rate), material stiffness, and the chemical



Figure 8-1. Ultrasonic welding machine.
(Courtesy Branson Ultrasonics Corp.)

makeup of the plastic being welded. Depending on compatibility regarding the factors just mentioned, certain dissimilar amorphous material combinations can successfully be welded together.

Energy Directors

Figure 8-2 shows a standard formula for determining the shape and size of a common “energy director” for sonic welding. The primary purpose of the energy director is to direct energy from the horn of the ultrasonic machine to the point of desired welding on the plastic part. The energy director focuses the ultrasonic energy to that point, thereby creating highly concentrated sonic energy, which causes the plastic to heat quickly. Without the energy director, the sonic energy would be dissipated over the entire mating surface area and be very weak, resulting in longer cycle times caused by the slower heating of the plastic.

Ease of Welding

Tables VIII-1 and VIII-2 indicate the relative *ease of welding* for some common thermoplastics. Ease of welding is a function of joint design, part geometry, energy requirements, amplitude, and fixturing. In addition, these

ratings are based on “near field” welding (the welding joint is located within 1/4 in. [6.35 mm] of the horn contact surface).

Note that the frequency used for all these types of welding will usually be at 20 kHz (80% of the time; 40 kHz may be required 20% of the time). Vibration welding, on the other hand, is normally performed at low frequencies, on the order of 250 to 300 Hz. Vibration welding is normally required when welding large components, such as automotive bumpers, or intricate units that may be damaged by higher frequencies. Products requiring hermetic sealing usually require vibration welding also.

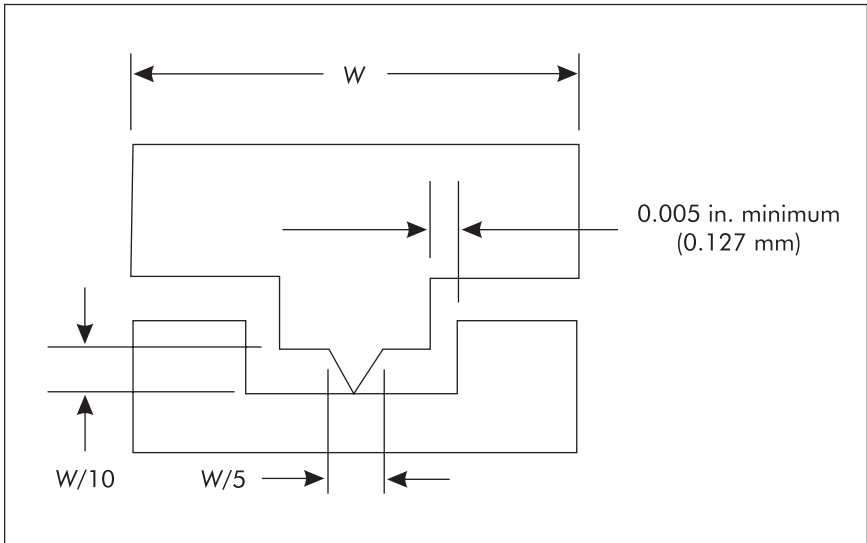


Figure 8-2. Energy director formula.

*Variables that Influence Weldability**

Polymer structure. Amorphous resins are characterized by a random molecular arrangement and a broad softening temperature (glass transition [T_g]) range that allows the material to soften gradually and flow without solidifying prematurely. These resins generally are very efficient with regard to their ability to transmit ultrasonic vibrations and can be welded under a wide range of force/amplitude combinations.

Crystalline resins are characterized by regions of orderly molecular arrangement and sharp melting (melt temperature [T_m]) and resolidification points. The molecules of the resin in the solid state are spring-like and internally absorb a percentage of the high-frequency mechanical vibrations of the ultrasonic generator. This makes it more difficult to transmit the ultrasonic energy to the joint interface, thus higher amplitude is usually required.

Melt temperature. Generally speaking, the higher the melt temperature of a resin, the more ultrasonic energy required for welding.

Stiffness (modulus of elasticity). The stiffness of the resin to be welded can influence its ability to transmit the ultrasonic energy to the joint interface. Generally the stiffer the material the better its transmission capability.

*This material provided by Branson Ultrasonics Corp.

Table VIII-1. Ease of Welding (Amorphous Materials)

Material	Standard Welding	Staking	Insertion	Spot Welding	Vibration Welding
ABS	E	E	E	E	E
ABS/polycarbonate blend	E-G	G	E-G	G	E
Acrylic (noncast)	G	F	G	G	E
Acrylic multipolymer	G	G	G	G	E
Butadiene-styrene	G	G	G	G	G
Polyphenylene oxide (PPO)	G	G-E	E	G	E-F
Polyamide-imide	G	N/R	N/R	N/R	G
Polyarylate	G	N/R	N/R	N/R	N/R
Polycarbonate	G	G-F	G	G	E
Polyetherimide	G	N/R	N/R	N/R	N/R
Polyethersulfone	G	N/R	N/R	N/R	N/R
Polystyrene (general purpose)	E	F	G-E	F	E
Polystyrene (rubber modified)	G	E	E	E	E
Polysulfone	G	G-F	G	F	E
PVC (rigid)	F-P	G	E	G-F	G
SAN-NAS-ASA	E	F	G	G-F	E
PBT/polycarbonate blend	G	F	G	G	E
Code: E = Excellent, G = Good, F = Fair, P = Poor, N/R = Not recommended					

Moisture content. Some materials, such as nylon, ABS, polycarbonate, and polysulfone are hygroscopic; that is, they absorb moisture from the atmosphere, which can seriously affect weld quality.

If hygroscopic parts are not sufficiently dried, when they are welded, the retained moisture will become steam. This trapped gas will create porosity (foamy condition) and often degrade the resin at the joint interface. This in turn degrades the potential of the material to produce a hermetic seal and creates poor cosmetic appearance (frostedness), degradation, and reduced weld strength. For these reasons, if possible, it is better that hygroscopic parts be welded directly from the molding machine to ensure repeatable results. If welding cannot be done immediately, parts should be kept dry, as-molded, by sealing them in polyethylene bags directly after molding. Then, drying the parts prior to welding can be done in special ovens; however, care must be taken to avoid material degradation.

Flow rates. Flow rate is the rate at which a material flows when it becomes molten, as indicated by the value obtained through the melt index test (ASTM

Table VIII-2. Ease of Welding (Crystalline Materials)

Material	Standard Welding	Staking	Insertion	Spot Welding	Vibration Welding
Acetal	G	G-F	G	F	E
Cellulosics	F-P	G	E	F-P	E
Fluoropolymers	P	N/R	N/R	N/R	F
Ionomer	F	N/R	N/R	N/R	N/R
Liquid crystal polymers (LCP)	F	G-F	N/R	N/R	N/R
Nylon	G	G-F	G	F	E
Polyethylene terephthalate (PET)	G-F	N/R	N/R	N/R	N/R
Polybutylene terephthalate (PBT)	N/R	N/R	N/R	N/R	N/R
Polyetheretherketone (PEEK)	F	N/R	N/R	N/R	G
Polyethylene	F-P	G-F	G	G	G-F
Polymethylpentene	F	G-F	E	G	E
Polyphenylene sulfide (PPS)	G	P	G	F	G
Polypropylene	F	E	G	E	E
Code: E = Excellent, G = Good, F = Fair, P = Poor, N/R = Not recommended					

#D-1238). Different grades of the same material may have different flow rates. Such differences may result in the melting of one component of an assembly and not the other. Thus a melt or flow is created, but not a solid bond. Consult the resin specifications to ensure compatibility of flow rates.

Mold release agents. Often called *parting agents*, mold release agents are applied to the surface of the mold cavity to provide a release coating to facilitate removal of the parts. Agents such as carnauba wax, zinc stearate, aluminum stearate, fluorocarbons, and silicones can be transferred to the joint interface and interfere with surface heat generation and fusion, inhibiting welding. If use of release agents is absolutely necessary, the “paintable” or “printable” grades should be used. These will cause the least amount of interference with ultrasonic assembly.

Plasticizers. Plasticizers are high-temperature boiling organic liquids or low-temperature melting solids added to resins to impart flexibility. Flexibility is achieved through these materials’ ability to reduce the intermolecular attractive forces of the polymer matrix. Plasticizers can also interfere with a resin’s ability to transmit vibratory energy. Attempting to transmit ultrasonic vibrations through

a highly plasticized material (such as vinyl) is like transmitting energy through a sponge. The energy is absorbed rather than directed to a focal point.

Flame retardants. Flame retardants are added to a resin to inhibit ignition or modify the burning characteristics. These chemicals are generally inorganic oxides or halogenated organic elements, and for the most part are not weldable. Typical retardants are aluminum, antimony, boron, chlorine, bromine, sulfur, nitrogen, and phosphorous. The amount of flame retardant material required to meet certain test requirements may vary from 1 to 50% or more by weight of the total matrix, thus reducing the amount of available weldable material. This reduction must be compensated for by modifying the joint configuration to increase the amount of weldable material at the joint interface and by increasing ultrasonic energy levels.

Regrind. Characteristics of the molded part may be adversely affected by a potential lower melting temperature of regrind. Therefore, control over the volume and quality of regrind is necessary. In some cases the use of 100% virgin material may be required to obtain the desired results.

Colorants. Although most colorants, either pigments or dyestuffs, do not interfere with ultrasonic assembly, a pigment loading of more than 5% can inhibit weldability. An application evaluation should be performed to determine feasibility and welding parameters for various pigments and loadings.

Resin grade. Different grades of the same resin can have a significant influence on weldability, owing to potential differences in melt temperatures and molecular weights among the grades. A general rule of thumb is that both materials to be welded should have similar molecular weight, and melt temperatures should be within 40° F (22° C) of each other.

Fillers. Fillers actually enhance the ability of some resins to transmit ultrasonic energy by imparting higher rigidity (stiffness). Common fillers such as calcium carbonate, kaolin, talc, alumina trihydrate, organic fillers, silica, calcium metasilicate, and micas can increase the weldability of the resin considerably with loadings of up to 35%. Above that level, there may not be enough resin at the joint surface to obtain reliable hermetic seals.

Even with lower percentage loadings, abrasive fillers can cause excessive wear on the surface of the ultrasonic horn. In this situation the use of hardened steel or carbide-faced (coated) titanium horns is recommended.

Reinforcements. The addition to resins of continuous or chopped fiber strands of glass, aramid, carbon, etc., can improve a plastic's weldability; however, rules of good practice governing the use of fillers should be observed. Long fibers can collect and cluster at the gate area during molding and be forced through in lumps rather than uniformly dispersed. This agglomeration can lead to an energy director containing a much higher percentage of reinforcement material as compared to resin. If this occurs, no appreciable weld strength can

be achieved since the energy director would simply imbed itself into the adjoining surface, not providing the required amount of molten resin to cover the joint area. This problem can be minimized or eliminated by using short fiber reinforcements.

Hot Gas Welding

Rigid PVC (vinyl) is the major common material assembled using the hot gas welding method. Other popular materials that can be used successfully are ABS, ABS blends, acrylics, polyethylene, polypropylene, polystyrene, and polycarbonate. Normally, filled materials are not acceptable for hot gas welding, but reinforced versions enjoy some success.

The process shown in Figure 8-3 is similar to metal welding in that a welding rod composed of the same material as that being welded is placed along a beveled joint area. Heat is then applied to that area by way of a hot gas (usually air, but nitrogen is recommended). The hot gas (400 to 1000° F [204 to 538° C]) melts the plastic to be joined as well as the welding rod. The gas tool continues on for further welding, and the plastic material in the heated area cools to resolidify and form a strong welded bond.

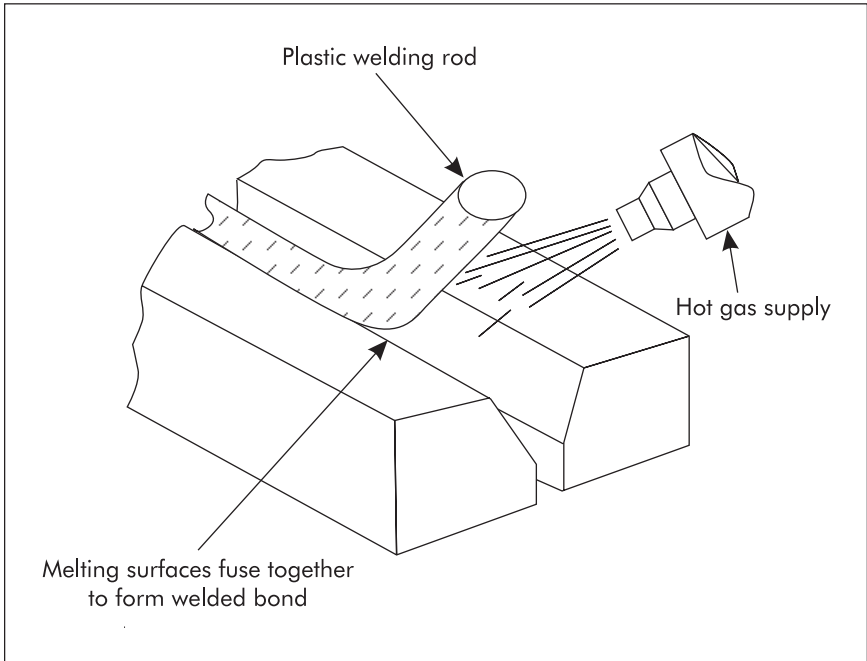


Figure 8-3. Hot gas welding process.

Induction (Electromagnetic) Bonding

The induction welding process consists of activating an electrodynamic field to excite a conductive bonding agent (such as metal screening or wire strands), thereby creating heat in the agent. This heat is absorbed by any plastic components that surround the bonding agent, resulting in the plastic melting. The melting plastics fuse to each other (and to the bonding agent) and solidify when the electrodynamic field is deactivated. Figure 8-4 depicts the process.

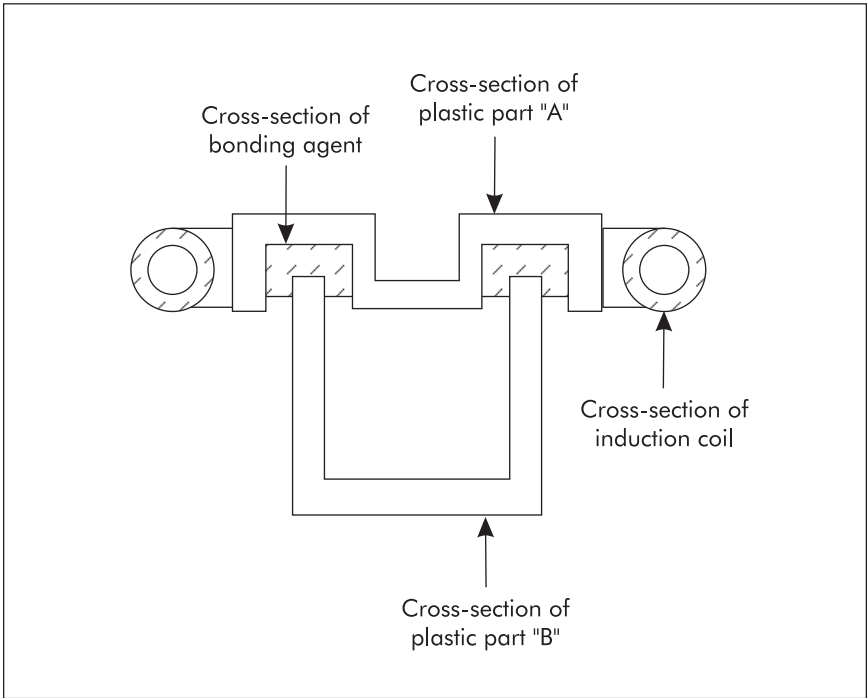


Figure 8-4. Induction welding (electromagnetic) concept.

A slight pressure is usually applied to the components being welded together, and the total process will take between 1 and 10 seconds, depending on the size of the area being welded.

Dissimilar materials can be welded together as long as they are thermally compatible to within 40° F (22° C) of their respective melting points.

A disadvantage to this process is that the conductive bonding agent remains sealed within the final welded part and becomes integral to that final part. Therefore, a fresh bonding agent must be used for each welded assembly.

Spin Welding (Friction Welding)

In the spin welding process (Figure 8-5), two parts are brought together with one of the parts spinning at a speed of between 100 and 1000 revolutions per minute (rpm). Slight pressure is applied as the two parts are brought together.

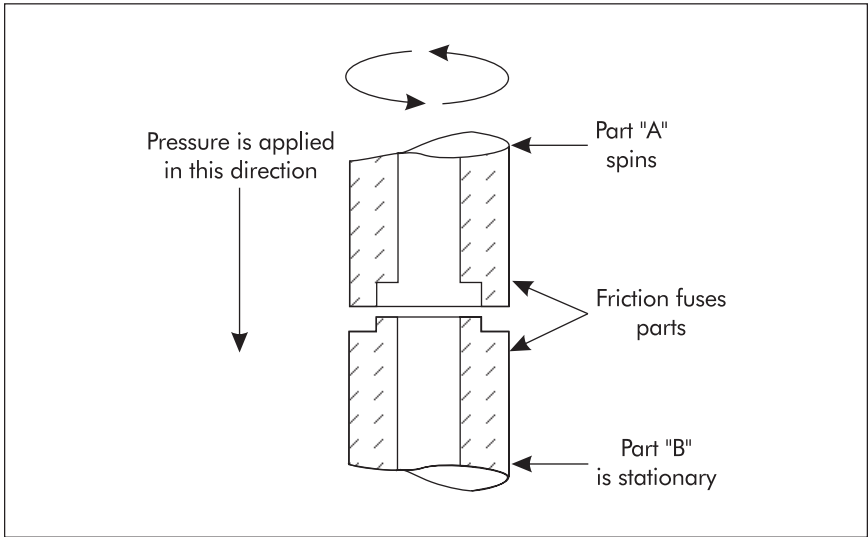


Figure 8-5. Spin welding.

Special equipment can be built to perform this operation that allows complete control of parameters such as rpm, pressure, contact speed, and dwell. However, it may be possible to perform adequate welds using common shop equipment such as a drill press or lathe.

Typical cycle times are in the range of 1 to 2 seconds. Although most rigid thermoplastics can be welded using this process, the softer materials such as low-density polyethylene create control problems and the welded area may be spongy and weak.

A disadvantage of the spin welding process is that normally the parts must be cylindrical in shape. However, it is possible to design a circular welding ring on noncircular parts and align the parts properly to create friction on the circular ring.

Adhesive Bonding

Thermoplastic materials may be bonded using monomers and solvents while thermoset materials can be bonded only by elastomerics and epoxies.

Monomer cements contain a specific plastic material that must be catalyzed to produce a bond. This can be done through heat, liquid catalyst, or UV light.

Solvent cements actually attack the surface of the plastic material in a controlled fashion and dissolve it, causing a molecular interlocking, after which the solvent evaporates.

Elastomeric adhesives contain natural synthetic rubber in a water- or solvent-based solution. They cure at room temperature (faster at elevated temperatures) and attain full bonding when the solvent or water is evaporated.

The epoxy adhesives (and similar-acting polyester and phenolic adhesives) act by forming a thermosetting layer of material between the two plastic surfaces to be bonded. This layer is catalyzed, causing it to cure and bond with the base layers.

Note: polypropylene, polyethylene, and the fluorocarbons are extremely difficult to bond adhesively due to their superior chemical resistance. It is considered better to bond these materials mechanically.

Joint design is instrumental to effecting a proper adhesive bond. Joints that combine both shear and tensile strengths are preferred. Although butt joints are sometimes successful, other designs such as lap joint, v-joint, and tongue-and-groove joints should be used to ensure proper bonding of all plastics. These are depicted in Figure 8-6.

Table VIII-3 shows those adhesives best suited for some of the more popular plastic materials.

MACHINING OPERATIONS

Machining operations include drilling, tapping, lathe turning, milling, sanding, and to some extent, grinding. Any secondary machining process removes some of the surface material of the plastic product being machined, and therefore reduces the hardness and luster of that surface. This must be taken into account, especially for products that will be exposed to severe environments or that require a high luster for aesthetic purposes. In the case of thermosets, and some thermoplastics, the luster usually may be restored by polishing the surface with mineral oil.

In most cases, plastics can be machined accurately if the proper tools, speeds, feeds, and coolants are utilized. Thermoplastic materials are especially susceptible to cutting speeds and cutter shape because of the heat generated by the machining process. If excessive, this heat will cause deformation, or even melting, of the plastic product being machined.

Drilling and Tapping Thermoplastics

Carbide drills are most suitable for drilling thermoplastics, but if they are not readily available, carbide-tipped or diamond-tipped drills can be used. If

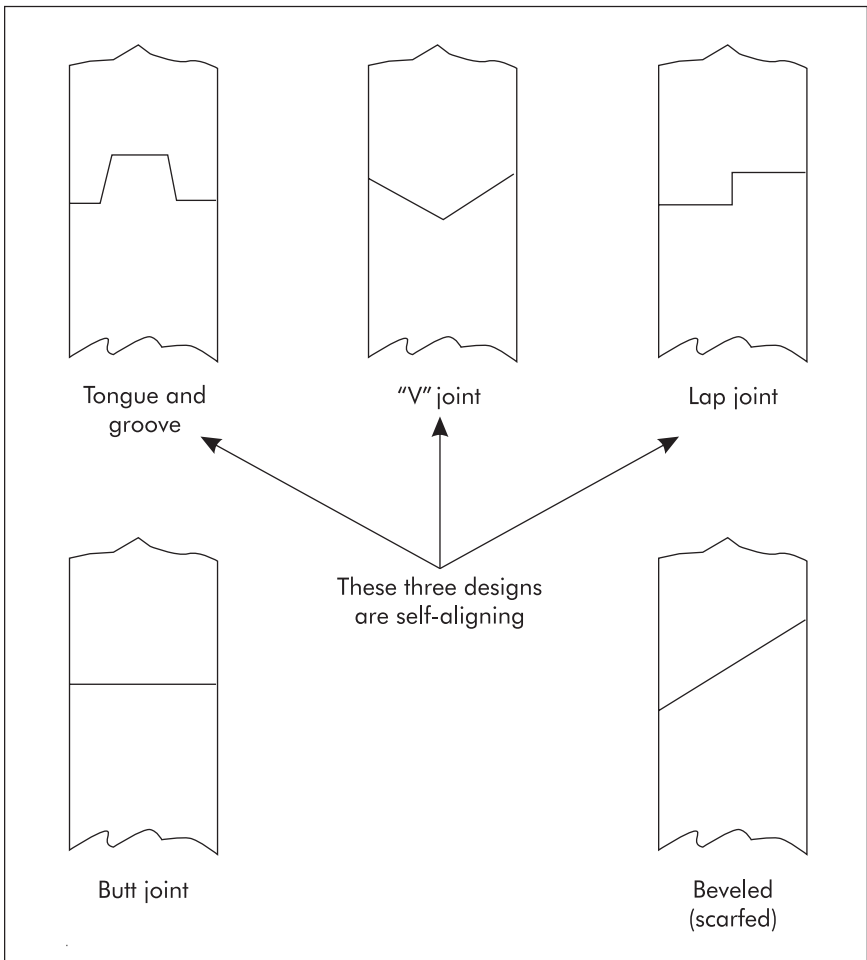


Figure 8-6. Adhesive bond joint examples.

surface finish is critical and a “mirror” finish is required, use of a diamond-tipped drill is mandatory.

Flutes should be highly polished and the drill cutting surfaces should be chrome plated or nitrided to reduce wear and increase cutting efficiency. Details of drill dimensions are shown in Figure 8-7.

The drill land (L) should be $1/16$ in. (1.59 mm) or less. The helix angle (HA) should be 30 to 40° . The point angle (PA) should be 60 to 90° for small drills (up to $1/8$ -in. [3.18-mm] diameter) and 90 to 115° for large drills (over $1/8$ -in. diameter). And, the lip clearance angle (LCA) should be 12 to 18° .

Table VIII-3. Adhesives for Bonding Common Plastics

Plastic Resin	Adhesive Type
Acetals	1, 2, 10, 22, 27, 30
ABS	1, 2, 3, 8, 10, 12, 18, 21, 22, 24, 27, 30
Cellulose acetate	1, 2
Epoxy	2, 5, 8, 10, 20, 21, 22, 24, 26, 27, 30
Melamine	4, 6, 8, 10, 16, 27, 30, 32
Nylon	1, 2, 6, 7, 10, 12, 17, 22, 24, 30
Phenolic	2, 5, 6, 8, 9, 10, 17, 20, 21, 22, 23, 24, 26, 27, 28, 29, 31, 32
Polycarbonate	1, 2, 3, 8, 10, 12, 24
Polyester (TS)	2, 3, 8, 10, 18, 20, 21, 22, 24, 27, 30
Polyethylene	8, 10, 17, 22, 24, 27, 29
Polyethylene terephthalate (PET)	2, 5, 9, 16, 17, 21, 22, 26
Polyimide	8, 9, 10, 11, 22, 25, 26, 27, 28, 29
Polyphylene oxide (PPO)	2, 3, 8, 10, 21, 22, 24, 27, 30
Polypropylene	10, 22
Polystyrene	1, 3, 12, 17, 18
Polysulfone	22
Polyurethane	10, 22, 24, 27
Polyvinyl chloride (PVC)	1, 2, 8, 10, 12, 16, 18, 21, 22, 24, 27, 30
Silicone	2, 25, 26
Adhesive Code	
Thermosets	Thermoplastics
1. Cyanoacrylate	13. Cellulose acetate
2. Polyester + isocyanate	14. Cellulose acetate butyrate
3. Polyester + monomer	15. Cellulose nitrate
4. Urea formaldehyde	16. Polyvinyl acetate
5. Melamine formaldehyde	17. Polyamide
6. Resorcinol formaldehyde	18. Acrylic
7. Phenol formaldehyde	
8. Epoxy + polyamine	
9. Epoxy + polyanhydride	
10. Epoxy + polyamide	
11. Polyimide	
12. Acrylate acid ester	
Elastomers	Alloys
19. Natural rubber	28. Epoxy phenolic
20. Butyl	29. Epoxy nylon
21. Polyisobutylene	30. Phenolic neoprene
22. Nitrile	31. Phenolic polyvinyl butyral
23. Styrene butadiene	32. Phenolic polyvinyl formaldehyde
24. Polyurethane	
25. RTV silicone	
26. Silicone resin	
27. Neoprene	

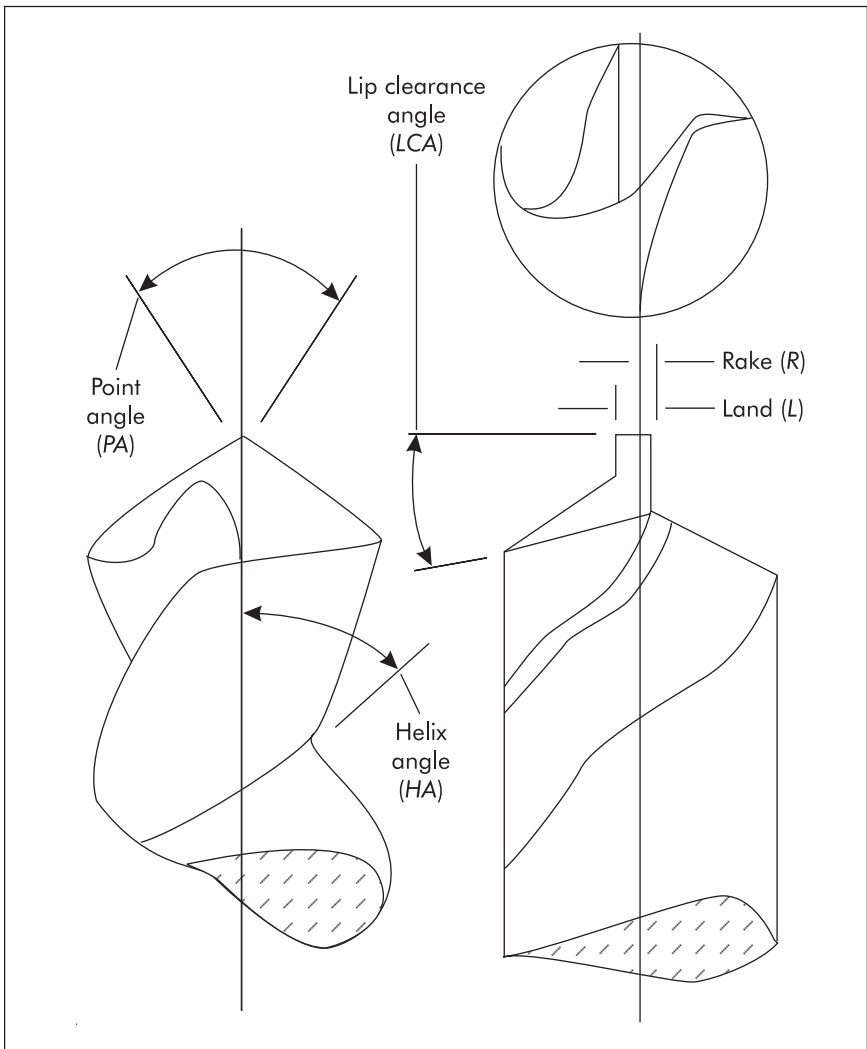


Figure 8-7. Drilling point details.

Drill feed should be approximately 0.005 in. (0.127 mm) per revolution of drill bit. Drill speeds should range from 5000 rpm for drill diameters up to 1/8-in. (3.18-mm) diameter, to 1000 rpm for drill diameters of 0.5 in. (12.7 mm) or greater, and prorated speeds for drills between these diameters. Slower speeds may be required on certain materials such as PVC to prevent overheating.

Taps for thermoplastics should be of two- or three-flute design and be made of solid carbide or chrome-plated (or nitrided) high-speed steel. Because of the

resilient tendency of some of the less rigid plastics, it may be necessary to use a tap that is slightly oversized (0.001 to 0.005 in. [0.025 to 0.127 mm] over nominal) which will result in proper sizing and thread depth after machining.

Drilling and Tapping Thermosets

Because of the inherent abrasive nature of thermoset materials, it is recommended that carbide drills be utilized for drilling operations. Carbide-tipped or diamond-tipped drills are also acceptable.

Flutes should be highly polished and the drill cutting surfaces should be chrome-plated or nitrided to reduce wear, reduce friction, and increase cutting efficiency.

Refer to Figure 8-7 for visual definitions of the following details:

- The drill land (*L*) should be 1/16 in. or less. The helix angle (*HA*) should be between 15 and 30°. The rake (*R*) should be 0 to 3°. The point angle (*PA*) should be 90 to 115°. And, the lip clearance angle (*LCA*) should be 12 to 18°. Because of the abrasive nature of the plastic materials, it is advisable to use drills that are 0.001 to 0.002 in. (0.025 to 0.051 mm) oversize.
- Drill feed should be approximately 0.005 in. (0.127 mm) per revolution of the drill bit. Drill speeds should range from 5000 rpm for drill diameters up to 3/32 in. (2.38 mm), to 750 rpm for drill diameters of 1/2 in. (12.7 mm) or greater, and prorated for drills between those diameters. Higher speeds will result in improved finishes but will reduce drill life due to wear.
- Taps for thermosets should be two- or three-fluted design and be made from solid carbide or chrome-plated (or nitrided) high-speed steel. Due to abrasiveness of the plastic it is recommended that 0.001 to 0.003 in. (0.025 to 0.076 mm) oversize taps be utilized.

Reaming Thermoplastics and Thermosets

Both thermoplastic and thermoset products may utilize reaming operations for accurate sizing of hole diameters. Reamers should be of fluted design for best surface finish and be made of carbide steel to minimize wear. For thermoplastic materials it is best to use reamers that are 0.001 to 0.002 in. (0.025 to 0.051 mm) oversize because of the resilient tendency of the material.

Reamer feeds and speeds should approximate those of the drilling operations mentioned earlier, and water-soluble coolants should be used to achieve the best surface finish and minimize effects of generated heat due to friction.

Turning and Milling Thermoplastics and Thermosets

Lathe and mill cutters should be tungsten carbide or diamond-tipped with negative back rake and front clearance (see Figure 8-8).

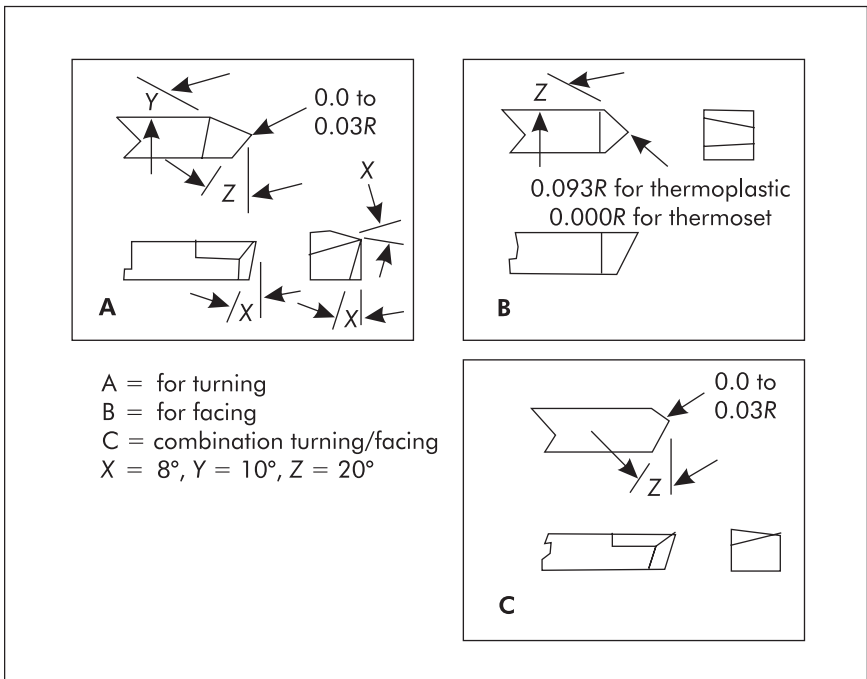


Figure 8-8. Design of turning points.

Tool bit designs should incorporate an X angle of 8° , a Y angle of 10° , and a Z angle of 20° .

Feeds should be between 0.01 and 0.02 in. (0.25 to 0.51 mm) per revolution. Speeds should be between 200 and 700 ft (61 and 213 m) per minute for thermoplastics and 1200 and 1800 ft (366 and 549 m) per minute for thermosets. Use of water-soluble coolants will improve surface finish and reduce point wear.

AUTOMATED SHAPE CUTTING

Water Jet

Perhaps the most popular automated cutting process currently used throughout all industries, water-jet technology utilizes the force of a thin stream of water under pressures in the range of 20,000 to 50,000 psi (137,900 to 344,750 kPa) to create a powerful cutting “point” that pierces plastic material cleanly and effortlessly. Dust and chips are nonexistent with water-jet cutting, and the addition of abrasive material to the stream allows cutting of the most difficult materials available today.

Although most water-jet cutting is performed on flat sheet stock, computer controlled three- and five-axis machines are capable of cutting shapes on very complex surfaces. Some units combine water-jet cutting with more conventional mechanical cutting processes to provide a variety of machining operations at a single station.

Laser

Laser cutters are used for cutting plastics when a fine polished finish is required, such as the edges of an acrylic sign. The laser unit cuts by focusing its concentrated beam at the exact point of required cutting, which causes the plastic to melt, vaporize, and solidify, resulting in an ultrasmooth finish. Laser cutting can be compared to laser printing on paper. Instead of thermally leaving an inked imprint, the laser cutter leaves a freshly melted and solidified plastic section.

Proper adjustments of laser energy and cutting speeds are critical to a successful cutting operation, and improper adjustments can cause the plastic to char, burn, or disintegrate. One result can be the release of toxic fumes: adequate exhaust venting is imperative.

Although most lasers operate effectively in the power range of 200 to 500 watts, some plastics require higher power cutters and equipment is now available with lasers operating in the 1000-watt and higher range.

SURFACE FINISHES AND DECORATING PROCEDURES

Preparation of Surface

Products that require postmold decorating require surface preparation to ensure adequate bonding of the decorative material to the plastic material. This surface preparation can range from a simple detergent wash to a complex acid bath, depending on the type of plastic being decorated and the decoration process selected. Some typical surface treatments beyond simple detergent washes are described in the following paragraphs.

Flame Treatment

Flame treatment is the most common method of preparing polyolefin and acetal plastics for decorating. These plastics, being crystalline materials, are highly resistant to chemicals, paints, and inks because of the slippery nature of their molded surfaces. Flame treatment, which consists of passing the molded product through a flame, causes the surface of the plastic to oxidize, which makes it receptive to adhesion of paints, inks, and other decorating media. The procedure requires a fair degree of control to ensure that the surface is oxidized properly without degrading or charring the plastic, but with practice the process can be done well by hand, or automated for closer control of parameters.

Corona Discharge

Surface oxidation of plastic material also can be achieved through the use of a corona discharge process in which the plastic product is passed over an insulated metal cylinder beneath a high-voltage conductor. An electron discharge then takes place between the conductors and the cylinder and it strikes the surface of the plastic product passing through. A corona is formed which causes the plastic to oxidize on the surface, thus preparing it for decorative coatings.

Plasma Process

In this process, low-pressure air is directed through an electrical discharge and expanded into a closed vacuum chamber in which the plastic product is placed. While passing through this chamber, nitrogen and oxygen gases are partially dissociated from the air and, in their atomic state, react with the surface of the plastic material to alter its physical and chemical characteristics. This altered surface then readily accepts decorative coatings.

Acid Etch

Some plastics, such as polycarbonate and some grades of ABS, do not accept decorative finishes even after exposure to treatments such as those described. In these cases it may be necessary (or preferable) to obtain a mechanical bond between the plastic and the decorative coatings. Mechanical bonds can be achieved through use of an acid wash process that attacks the surface of the plastic and creates uniform, microscopic craters of exposed resin. These craters provide the surface texture necessary to physically capture the decorative coating and lock it to the plastic surface.

Applied Finishes

Painting

Paint can be applied to a plastic product through brushing, spraying, rolling, or dipping, either manually or mechanically. Most plastics are painted using the standard spray process, as shown in Figure 8-9.

Spray painting is referred to as a *line-of-sight* process, meaning that what is painted is only what can be seen by the spray gun unit.

Although most plastics can be painted, the success of the process depends on proper surface preparation and the use of specific paint formulations for specific plastics. These formulas must be adjusted to address the plasticizer (if any) in the plastic to make sure there is no migration into the paint; the heat distortion temperature of the plastic, which determines whether the paint should be air-dried or oven-baked; and the chemical resistance properties of the plastic to determine which solvent system should be used to ensure proper adhesion without crazing. In some cases water-soluble paints have found successful applications.

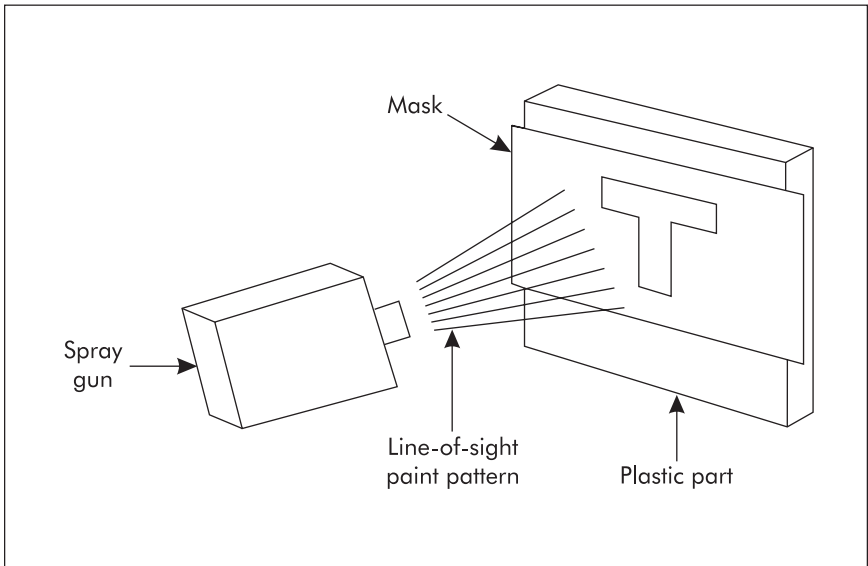


Figure 8-9. Spray painting.

Plating (Electroplating)

Electroplating plastic material requires that the normally nonconductive plastic be made conductive. That usually necessitates that a conductive base metal first be applied to an etched and sensitized plastic surface. Then a plating material can be applied to the conductive metal. Procedures vary depending on the specific plastic being plated, and detailed plating information is available from material suppliers of the specific plastic.

Metallic plating of plastics may be required for either decorative or functional purposes. Examples of decorative purposes would include escutcheons, plumbing fixtures, and jewelry finishes. Examples of functional purposes would include circuit board traces, electronic (EMI) shields, and corrosion-resistant surfaces.

Not all plastics can be plated, and those that can be usually require that a specific plating grade be used. Some of the more popular plastics that accept plating include ABS, acetal, acrylic, alkyd, cellulose, epoxy, phenolic, polycarbonate, polyurethane, TFE fluorocarbon, urea, and rigid PVC.

Vacuum Metallizing (Deposition)

When a plastic product needs a bright metallic finish but does not require the thick, rigid, protective plate created by electroplating, the less expensive method of vacuum metallizing should be given consideration.

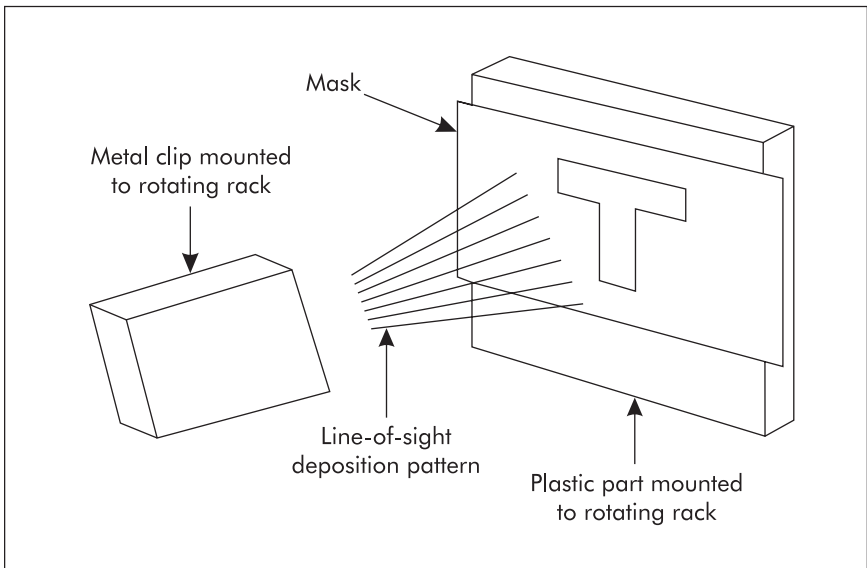


Figure 8-10. Vacuum metallizing process.

This deposition process requires that the plastic product be coated with a lacquer *base coat* to minimize surface defects and increase adhesion properties of the plastic substrate. The product(s) is then placed on a rack inside a vacuum chamber, along with small clips of the metal to be deposited, as illustrated in Figure 8-10.

During operation the metal clips are electrically heated to the point of vaporizing, at which point, due to the high vacuum in the chamber, the vaporized metal is deposited on all line-of-sight surfaces. The product is then removed from the chamber and coated with another layer of lacquer to protect the metal finish, which is approximately only 5 millionths of an inch (5 microinches [0.127 microns]) thick. If a color tone is required, it is applied at the same time as the final lacquer.

*Hot Stamping**

There are three main methods of hot stamping: roll-on decorating, peripheral marking, and vertical stamping (see Figures 8-11 through 8-14).

Roll-on decorating. This process is ideal for applying foils or preprinted heat transfers to part surfaces with large areas. With this method, a silicone rubber roller applies heat and pressure to release the print medium onto the

*This information supplied by United Silicone, Inc.

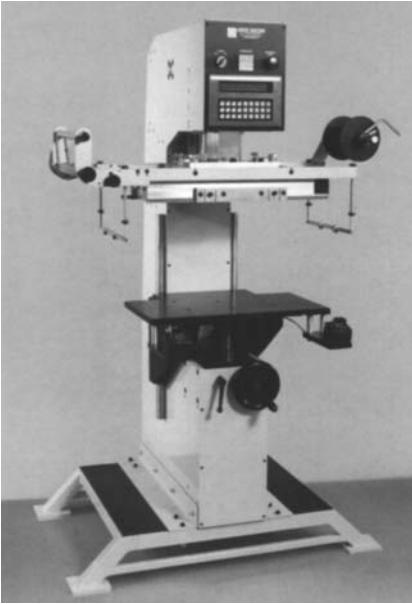


Figure 8-11. Typical hot stamp machine. (Courtesy United Silicone, Inc.)

plastic substrate. The advantage of this process is that the rubber roller material maintains line contact and pushes out trapped air between the printed medium and the decorating surface so that air bubbles are eliminated.

Peripheral marking. Used mainly for applying foils or preprinted heat transfers to the periphery of cylindrical or slightly conical parts, the peripheral marking method rolls the plastic product under a flat stamping die, or roller, that applies heat and pressure to release the print medium onto the plastic substrate. The advantage of this process is that up to 360° of the part circumference can be decorated in one machine cycle.

Vertical stamping. The most common hot stamping method, vertical stamping is ideal for applying foils or preprinted heat transfers to smaller areas of flat or slightly-crowned plastic products. It can also be used for up to a maximum of 90° on the circumference of cylindrical or spherical shaped products.

Typically, a silicone rubber die is mounted to the heated head of a vertical machine and positioned directly over the part to be decorated. The rubber die

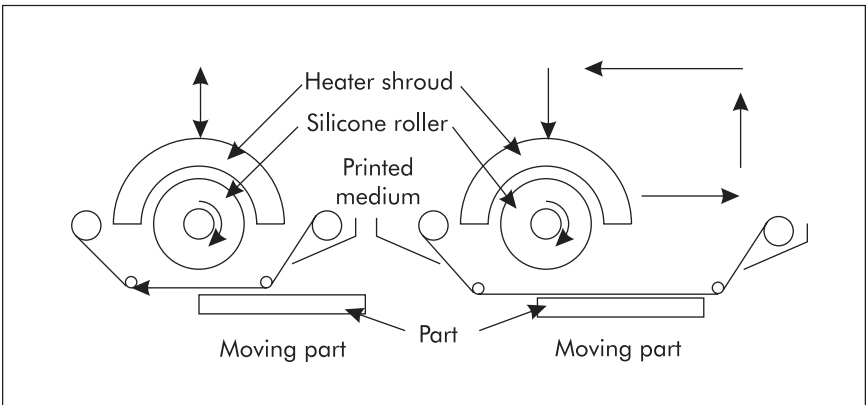


Figure 8-12. Common roll-on decorating method. (Courtesy United Silicone, Inc.)

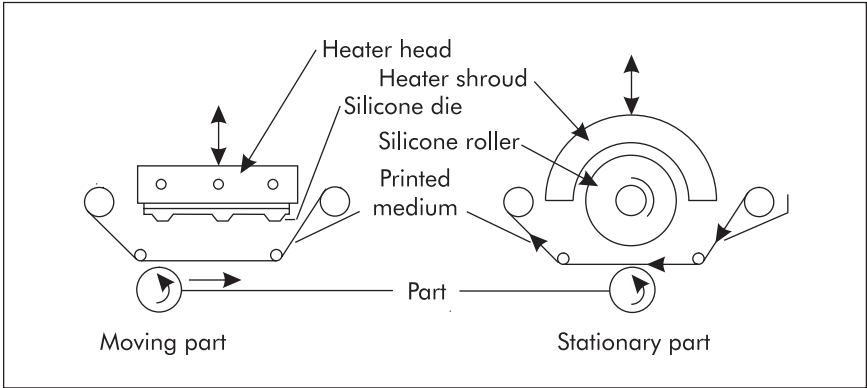


Figure 8-13. Common peripheral marking method. (Courtesy United Silicone, Inc.)

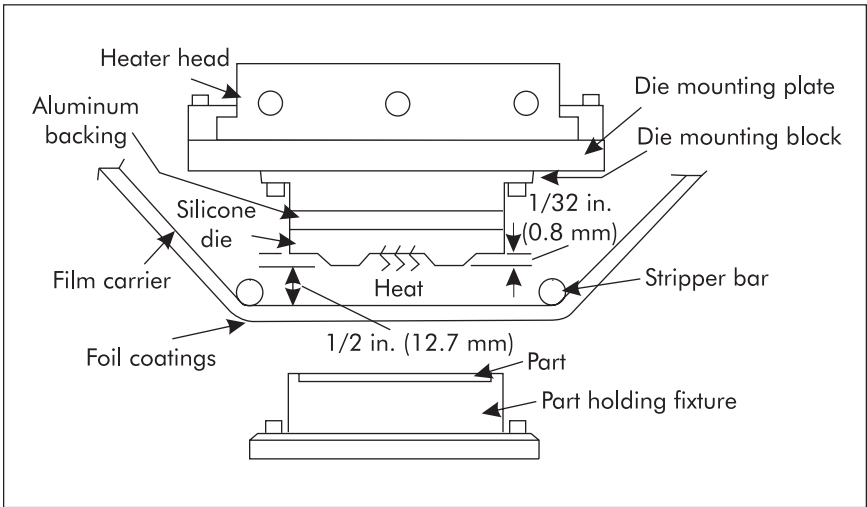


Figure 8-14. Typical vertical stamping method. (Courtesy United Silicone, Inc.)

contains raised graphics (approximately 0.031 in. [0.79 mm]), and is heated to the approximate melting point of the plastic to be stamped. Placed between the rubber die and the plastic product is the hot stamp foil, which consists of various thin coatings deposited on a film carrier. The rubber die is lowered and pushes the foil against the waiting plastic product. Heat from the die causes the release agents in the foil to activate and also softens the very surface of the plastic product. The decorative resins in the foil are transferred and thermally bonded to the plastic part.

*Pad Printing (Heat Transfer)**

Pad printing is usually performed in a manner similar to a printing press for paper (see Figure 8-15). A pad (usually made of silicone rubber) is inked with the image to be placed on the plastic product. This inking is performed by pressing the rubber pad onto a steel or nylon plate that has the image etched or machined into it and has ink screened into that image. The inked pad is then brought to the surface of the plastic product and pressed against it, transferring the inked image.

Screen Printing

In this process, ink or paint is forced through the mesh of a plastic or metal screen by pulling a squeegee across the screen which is placed against the surface



Figure 8-15. Typical pad printing machine. (Courtesy United Silicone, Inc.)

*This information supplied by United Silicone, Inc.

of the plastic product being decorated, as shown in Figure 8-16. An artwork mask is secured to the screen which causes the ink to flow through the mesh only in specific areas, thus forming the required design on the product. The artwork mask is in the form of a stencil made by placing the artwork positive on a photo-sensitive film and exposing the film to a light source that etches the image onto the film.

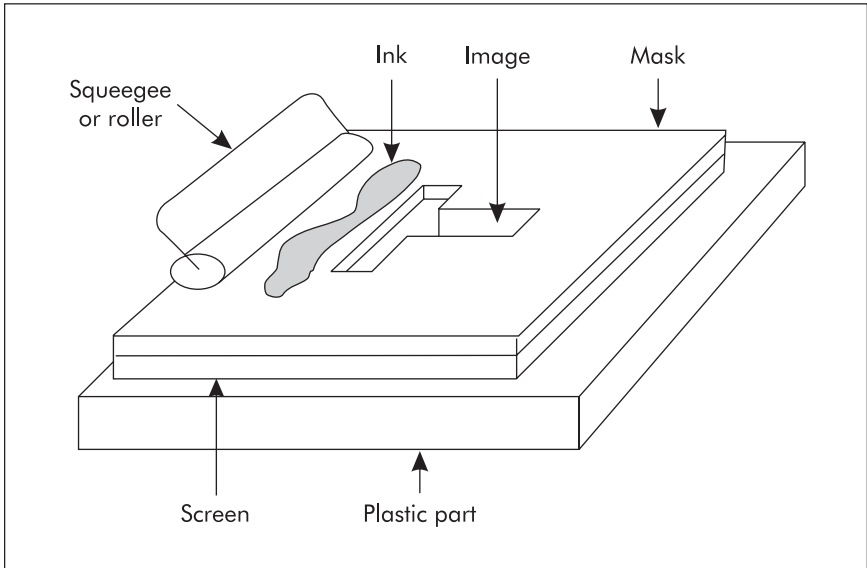


Figure 8-16. Screen printing process.

In-process Finishes

Molded-in Color

An alternative to painting a color on plastic products is to mold the color into the plastic to begin with. The advantages are, first, the color does not have to be applied as a secondary operation; second, the color does not wear off the surface during the product's life cycle.

Molded-in color is achieved by blending a coloring agent with the plastic pellets used in the injection molding process. Usually this agent is a powdered dye (dry color) or concentrate. Sometimes, liquids are used.

Dry color is mixed with the raw plastic pellets prior to injection molding, usually through use of an automatic blender mounted on the molding machine hopper. Sometimes, however, the dry colorant can be tumbled in with the raw pellets by hand mixing in a large container such as a clean metal barrel or cement

mixer. Dry colorants are usually shipped in small bags that are premeasured to be mixed with 50-, 100-, or 200-lb (23-, 45-, 91-kg) batches of raw pellets.

Color concentrates are produced by extruding heavy concentrations of colorant agent into a basic batch of plastic material compatible with the pellets that are to be colored. These heavy colored batches are then mixed with the basic pellets at approximately 5% ratio (5 lb [2 kg] concentrate to 100 lb [45 kg] pellets) and the mixture is injection-molded as a total blend.

Molded-in Symbols

For ease of decorating and low secondary costs, molded-in letters, symbols, and designs can be coated by using a rubber roller with ink or paint applied to its surface. The molded-in image is created by machining the image directly into the mold steel. This produces a raised image on the molded part, and this raised image receives a coating only on the top of the raised surfaces much like a rubber stamp against an ink pad.

In some cases, the molded-in image can be embossed—at higher cost—into the surface of the plastic product. For decorating these symbols the ink or paint is wiped into the image's recesses and then the surface is wiped clean leaving only the coating in the recesses.

Two-color (Two-shot) Molding

Products such as typewriter keys and computer keyboard keys are susceptible to surface wear due to constant use. Because of that, a process known as two-color molding (Figure 8-17) is used for creating the decorative finish, such as keyboard symbols. This process is actually an injection molding process performed twice.

In the first step, the base color material is molded into a basic shape. Then the second material is injection-molded into the remaining open spaces. On a keyboard, the key itself is molded in an off-white material, but the area where a symbol is intended is molded as an open space. That space is then filled during the second injection step with a material of a different color such as black. As the surfaces wear in use, they wear consistently so that the two colors are always apparent.

While two-color molding uses more expensive equipment than ink-stamping or painting, it is recommended when wear is a factor to be considered as a function of final product design.

Textured Surface

Textured surfaces can be painted onto a plastic surface, but in most cases they are molded directly into the product. This is accomplished by etching or machining the surface of the cavity image in the mold. The amount of texture depth and size is determined by preselecting a pattern from samples available from companies specializing in this type of work. One word of caution: when

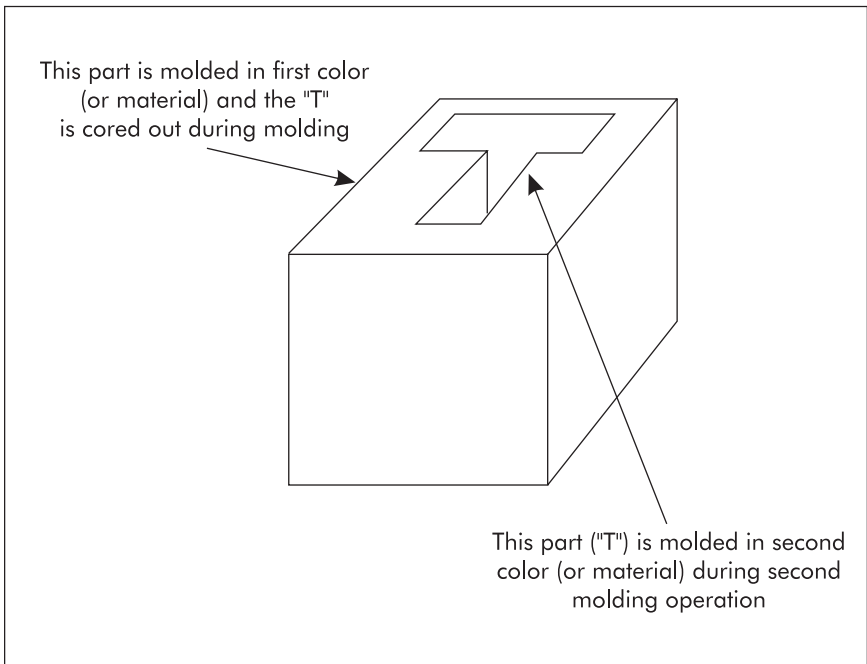


Figure 8-17. Two-color molding concept.

comparing samples, try to have the samples created in the same material and color which will be used in producing the molded product. A specific texture shown in a red ABS will take on a totally different appearance from that same texture in a black nylon.

Textures are available in hundreds (if not thousands) of variations ranging from a fine satin finish to a heavy alligator hide. They are usually used for aesthetic purposes only, although some heavy textures may even add structural strength to the molded product. They are also used to hide molding imperfections such as splay, knit lines, and blush.

The disadvantages of using textures are the original cost to create the texture in the mold, the fact that the texture will eventually wear off of the mold in high friction areas and must be replaced, and the problem of selecting a texture based on previously molded samples that are not exactly like the final product to be produced.

In-mold Overlays

For thermoset products, in-mold overlay decorating (Figure 8-18) is accomplished by placing in the mold a foil (overlay) consisting of a thermoplastic sheet that has the required image printed on it. This is then coated with a

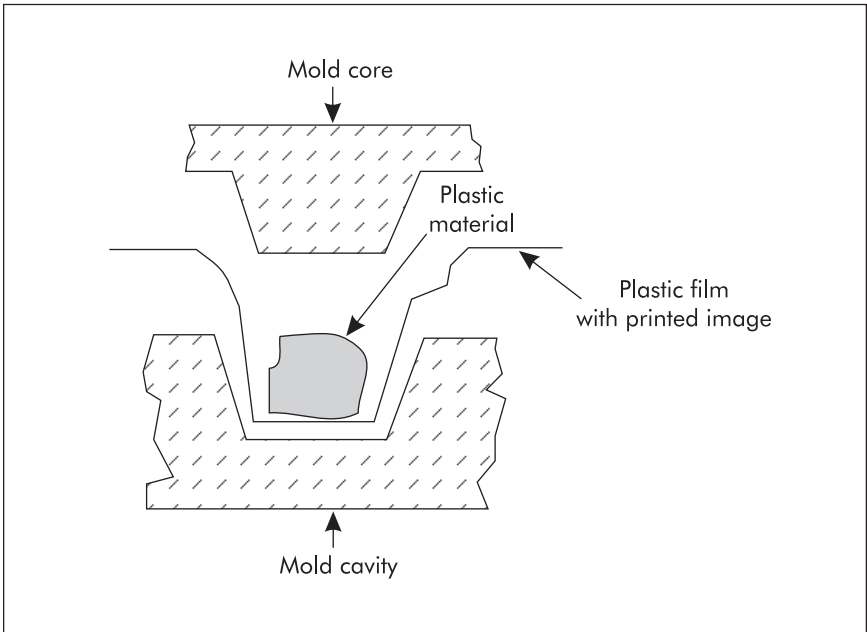


Figure 8-18. In-mold overlay process.

partially-cured thermosetting resin. During the compression molding process the entire overlay is fused to the curing molding compound and becomes one with the finished product.

For thermoplastic products, a similar process is used. But because the high pressures created during the injection process would simply tear apart the overlay, a slightly different approach is taken. This process, called *low-pressure molding*, incorporates injecting a charge of plastic material, under very low pressure, into an open mold. The mold is then closed and the plastic is compression-molded to create the final product. This keeps the fragile overlay from being destroyed.

A recent development in *controlled pressure* molding, allows thermoplastic materials to be injected at high pressures, but slowly, so that overlays placed in the mold do not wrinkle or tear. This process* results in reducing the injection pressure just before the mold cavities are filled, thus minimizing the amount of clamp tonnage required. This means that molds can be run in presses with much

*Patented by Hettinga Equipment, Inc.

smaller clamp units than for standard injection processes and the molded products will exhibit much less molded-in stress owing to the lower final injection pressures.

SUMMARY

A secondary operation can be defined as any operation performed on a product after it has been molded. Such operations normally include, but are not limited to, assembly, machining, and finishing (including decorative finishes).

Secondary operations should be considered when the annual volume requirements are less than 25,000 pieces, when tool costs are unexpectedly high, when production schedules may be jeopardized by time to build sophisticated molds, and when a labor-heavy environment exists.

Ultrasonic bonding processes can be used to weld a wide variety of plastics. Crystalline materials require greater amounts of energy than amorphous and are much more sensitive to joint design, horn design, and fixturing. Basically, the higher the melt temperature, the more ultrasonic energy required for welding. The major factors affecting weldability include polymer structure (amorphous versus crystalline), melt temperature, melt index (flow rate), material stiffness, and the chemical makeup of the plastic being welded.

Surface and decorative finishes are dependent on proper surface preparation of the plastic product being finished. Usually, a specific grade of plastic must be used, and the surface must be prepared using anything from a basic detergent wash to more sophisticated (and costly) etching processes.

There are two classifications of surface finishes: those applied during the molding processes, referred to as *in-process* finishes; and those applied after the molding process, referred to as *applied* finishes.

In-process finishes include molded-in color, molded-in symbols, two-color molding, textured surfaces, and in-mold overlays.

Applied finishes include painting, plating, vacuum metallizing, hot stamping, pad printing, and screen printing.

QUESTIONS

1. Define what is meant by a secondary operation.
2. List two circumstances in which secondary operations should be considered.
3. What is the normal range of sound waves created by the sonic welding process?
4. List two variables that influence the ultrasonic weldability of plastic products.

5. What adhesive could be used for successfully bonding polysulfone to nylon?
6. What is the range of drill speeds recommended for drilling most thermoplastics?
7. Why is surface preparation usually required before applying a finish to plastic parts?
8. List two common surface preparation treatments.
9. Define the difference between *applied* finishes and *in-process* finishes.
10. List three of the more common finishes and identify them as *applied* or *in-process* finishes.

Testing and Failure Analysis

9

DISTINGUISHING THE TERMS

Testing and failure analysis, insofar as we are using them here, should be defined separately. Testing is an activity performed to analyze the basic materials and product design concepts followed in the manufacture of a molded product. Failure analysis is performed on molded products that have failed to meet their intended design criteria, either shortly after being molded or in use by a consumer. We look at each of these subjects individually, although some of the same procedures and methods may be used for both. The procedures and methods shown are not all-inclusive; they are representative of the procedures and do not reflect exact methodology or principles. The source for most of the information that follows is the American Society for Testing and Materials (ASTM) book of standards.* In addition, we list the equivalent test standard number assigned by the International Organization for Standardization (ISO) which is becoming more accepted on a global basis.

TESTING

The type of testing we discuss requires calibrated equipment, well-documented procedures and results, proper training of test personnel, multiple test runs, and proper sample preparation. Proper preparation of the sample to be tested is paramount, since improper sample preparation can cause improper test results. In the following paragraphs we describe some of the more common tests performed. There are others, of course, and certain products and applications will require tests designed specifically for them.

Electrical Testing

Plastics make good insulators. Because they are considered nonconductive, they are used for such products as screwdriver handles and connector housings. These products depend on the property of the plastic to withstand exposure to electrical current, and testing procedures have been developed to monitor the capabilities

*Revised annually and available from American Society for Testing and Materials, Philadelphia, Pennsylvania 19103.

of the plastics to fulfill their mission. The common tests for electrical requirements are dielectric strength, dielectric constant, volume resistivity, surface resistivity, and arc resistance, as detailed in the following paragraphs.

Conditioning Samples

Electrical testing is dependent upon the proper moisture level being present in the plastic sample being tested. The sample must be conditioned, per ASTM test #D-618, to establish this level. The basic conditioning method, known as Procedure A, consists of placing the sample in a standard laboratory environment of 73° F (23° C) and relative humidity of 50% for a minimum period of 40 hours, for samples up to 0.275-in. (7-mm) thick. For samples over this thickness, the time is increased to a minimum of 80 hours. The samples must be tested immediately if removed from this conditioning environment. Preferably the testing can be performed while the samples are still in the environment.

Dielectric Strength

This test (ASTM #D-149 [ISO #IEC 243-1]) is designed to measure the amount of voltage required to arc through a specimen of the plastic being tested. Voltage,

starting at 0, is applied to one side of the specimen and increased until it arcs through the specimen (see Figure 9-1). The specimen can be a sample cut directly from a molded product or a flat sheet representing the same material and thickness as a proposed product.

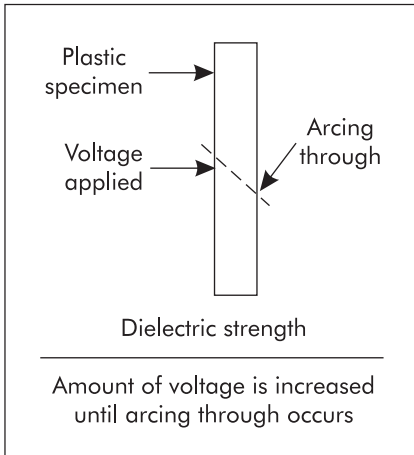


Figure 9-1. Dielectric strength testing.

Dielectric Constant

This test (ASTM #D-150 [ISO #IEC 250]) is designed to measure the electrical capacitance of a specific plastic cross-section as a ratio to a similar cross-section of air, as shown in Figure 9-2. The frequency range that can be covered extends from less than 1 Hz to several hundred megaHertz.

Volume Resistivity

This test (ASTM #D-257 [ISO #IEC 93]) is used to measure the ability of a plastic to resist an electric current through its bulk (see Figure 9-3). The test is used as an aid in designing electrical insulators.

Surface Resistivity

The surface resistivity test (ASTM #D-257 [ISO # IEC93]), Figure 9-4, is similar to that used for determining volume resistivity but is used to measure the

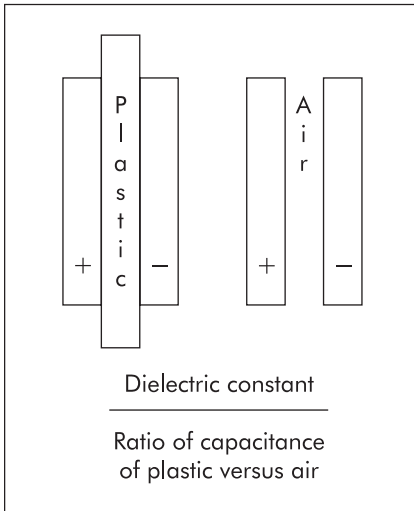


Figure 9-2. Dielectric constant testing.

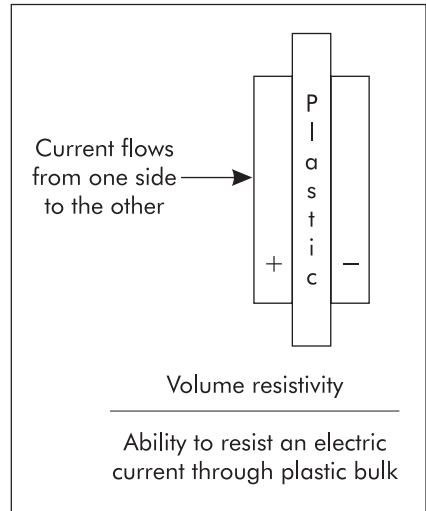


Figure 9-3. Volume resistivity testing.

ability of a plastic to resist current across its surface. This property is of importance when designing products such as circuit boards and connectors.

Arc Resistance

As depicted in Figure 9-5, this test (ASTM #D-495 [ISO #N/A]) is designed to measure the amount of time required for an electrical arc to carbonize the surface of a specific plastic specimen, thereby making it conductive. Although the results of this test are not generally used for specification purposes, they are utilized in the initial material selection process to aid in choosing one plastic over another.

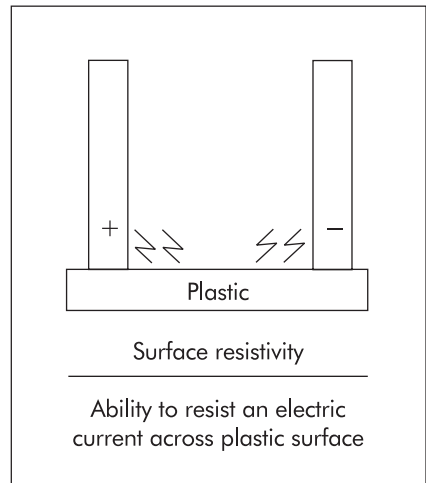


Figure 9-4. Surface resistivity testing.

Physical Testing

In this category are tests that determine the physical values of plastics in areas such as shrinkage rate, density (specific gravity), water absorption, moisture content, and melt flow index. These properties affect the ability of a specific plastic to be processed during injection molding and are instrumental in determining the final appearance of the molded product.

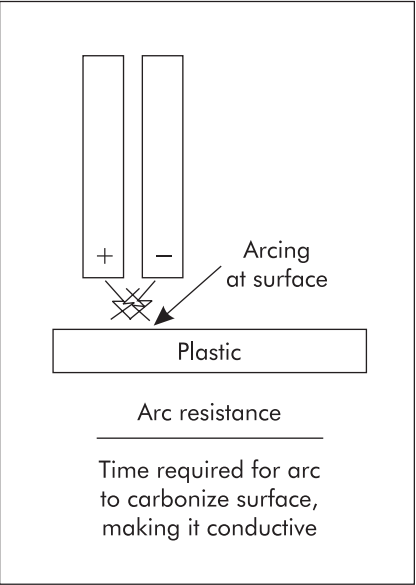


Figure 9-5. Arc resistance testing.

Shrinkage Rate

The shrinkage rate test (ASTM #D-955 [ISO #294-4]) is used to measure the amount of shrinkage (in./in. or mm/mm) that occurs in a specific plastic after it has been heated and injected into a mold, then allowed to cool (see Figure 9-6). Initial measurements are taken between 2 and 4 hours after removal from the mold and additional measurements are taken at approximately 20 hours and 44 hours after removal from the mold.

The amount of shrinkage must be measured both parallel with and across the direction of flow. Two different product designs are used to make those measurements, as shown in Figure 9-6. Many variables affect shrinkage, such

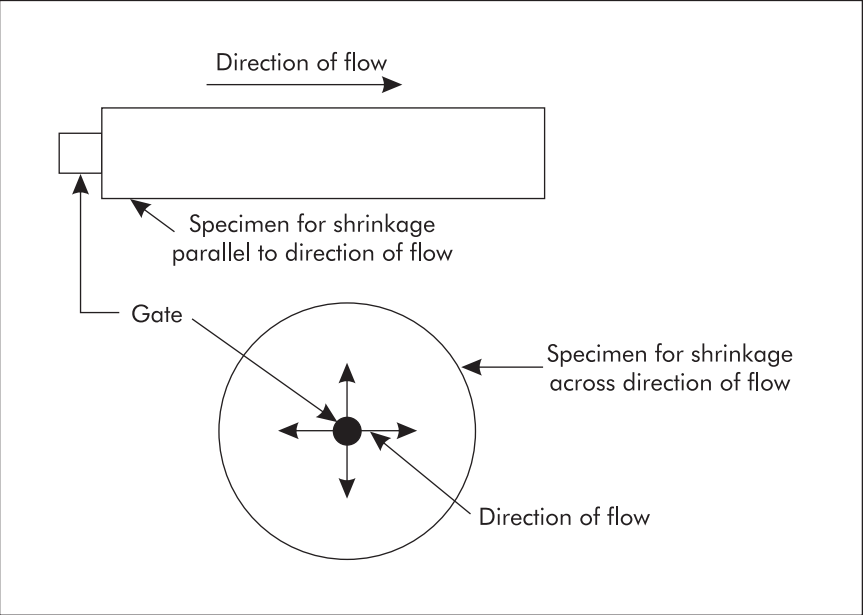


Figure 9-6. Shrinkage rate testing.

as the material temperature, flow rate, injection pressure, size of sprue and nozzle, percentage of reinforcement (if any), and other factors. For this reason, this test is to be used for reference purposes only.

Density

The density test (ASTM #D-792 [ISO #1183:1987]), Figure 9-7, is used to determine the weight of a specific volume of a particular plastic. The measurement is stated as grams per cubic centimeter (g/cm^3) and will range from approximately 0.95 to 1.60. As a reference, water is considered to have a density of $1.0 \text{ g}/\text{cm}^3$. A plastic with a value of less than 1.0 will float on water, and a plastic with a value of more than 1.0 will sink.

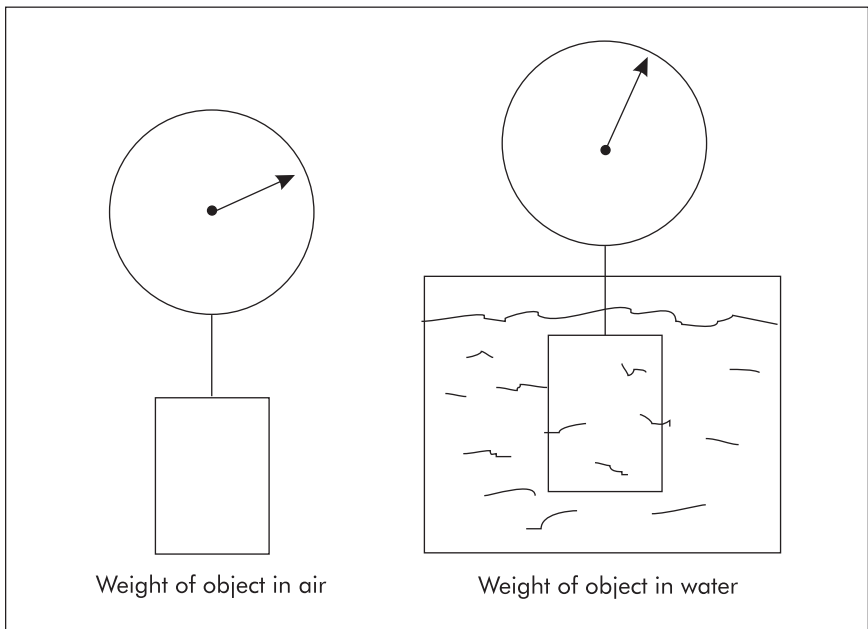


Figure 9-7. Determining the density of a plastic material.

Density is actually a measurement of the weight of 1 cm^3 of plastic part (or material). Because of this, density is closely related to specific gravity which compares the weight of a material in air to the weight of that same material in water. The common "water volume displacement" test, which is used for determining specific gravity, also can be used for determining density. Basically, this test measures the amount of water displaced by a given volume of plastic. First, the plastic piece is weighed in air by suspending it on a wire from a weighing

device. Then the plastic piece, while still suspended on the wire, is weighed again while placed in a container of water. A ratio is established which shows the weight of the object in air divided by the weight of the object in air minus the weight of the object in water. This is shown as:

$$\text{Density} = \frac{A}{A - B}$$

Water Absorption

This test (ASTM #D-570 [ISO #62:1980]) is used to measure the amount of moisture absorbed by a plastic material over a period of generally 24 hours, but may require measurements for a period of weeks depending on the specific plastic being tested.

As illustrated in Figure 9-8, properly sized and conditioned samples are weighed and placed in a container that allows them to be totally immersed in water. After 24 hours the samples are removed, surface moisture is removed, and the samples are weighed again. The amount is expressed as a percentage of weight increased by absorption of water. For materials that absorb high levels of moisture the test can be repeated by immediately re-immersing the samples in water. Measurements can be taken every 24 hours until the amount of increase is less than 0.002 oz (5 mg). At that point the sample can be considered completely saturated.

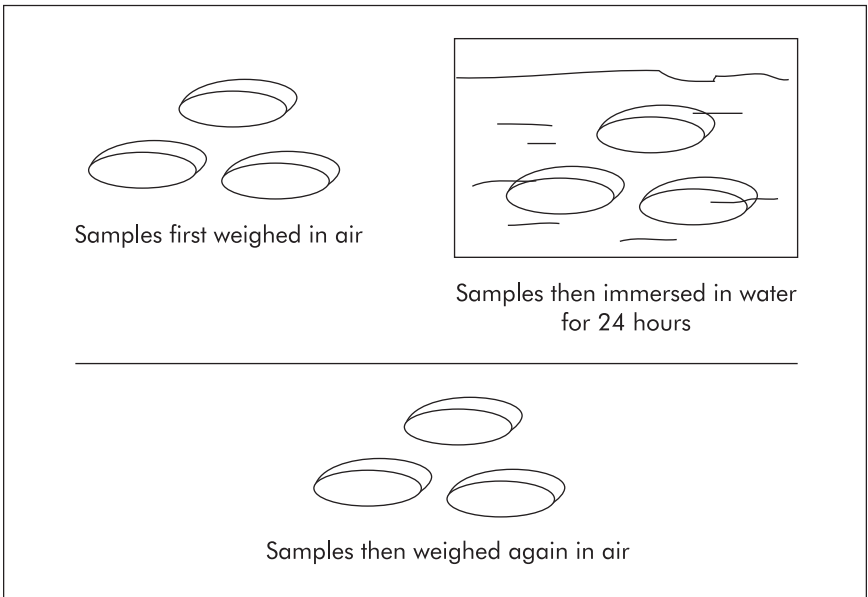


Figure 9-8. Water absorption measurement.

Moisture Content

This test (TVI [Tomasetti Volatile Indicator], ASTM-N/A, ISO-N/A) is named after the General Electric Plastics Section application engineer who developed the technique. It is an inexpensive, accurate method of determining the presence of moisture. Sometimes called the *resin dryness test*, the TVI requires only a hot plate capable of maintaining 525° F (274° C) temperature, two glass laboratory slides, tweezers, and a wooden tongue depressor.

The test is simple to perform. Two glass slides are placed flat on a hot plate, which is set to maintain a temperature of 525° F (274° C). When the hot plate has reached temperature and is thermally stabilized, the tweezers are used to deposit two or three plastic pellets on one of the slides. The second slide is immediately placed on top of the first, creating a sandwich of two slides with the pellets between them. A tongue depressor is then pressed against the sandwich of slides and pellets until the pellets melt and flatten out into 0.5-in. (12.7-mm) circular patterns (see Figure 9-9a).

Figure 9-9b shows typical results of the TVI test. The patterns on the left represent evidence of moisture. The moisture results in bubbles being formed in the melting resin. If only one or two bubbles are present, the indication is one of trapped air rather than moisture. The patterns on the right represent dry material containing no moisture. This material would be acceptable to mold as is, if used within an hour or two.

Melt Flow Index

This test (ASTM #D-1238 [ISO #1133:1991]) is used to determine the processability of a specific plastic. It also can be utilized to establish the batch-to-batch consistency of material as it is provided by the material supplier.

The test is performed, as shown in Figure 9-10, by placing material (regrind or virgin) in the preheated barrel of the machine. The barrel is heated to a specific temperature, depending on the plastic being tested. Then a weight is placed on the end of the plunger rod which causes the plunger to move forward. The amount of plastic that exudes from the nozzle over a 10-minute period is measured. The test value is expressed as “grams per 10 minutes,” and will usually range between 4 and 20 depending on the flowability of the specific grade and family of plastic being tested.

The primary value of this test is as an incoming material quality control. A value can be established as ideal for a specific product. That value can then be requested of the material supplier for all future material purchases, and a letter of certification can be requested of the supplier for each shipment. When the material arrives, it can be tested for *melt index value*. If the value is out of range, the material can be returned at the supplier's expense, or used as long as it is understood that material properties may be affected. Table IX-1 shows examples of how properties may be altered by a value that is lower than requested.

Note: permeability and gloss actually *decrease* as the melt index value drops.

Mechanical Testing

Mechanical testing is performed to determine structural properties, such as tensile, compression, flexural, creep, and impact strengths, of a specific plastic

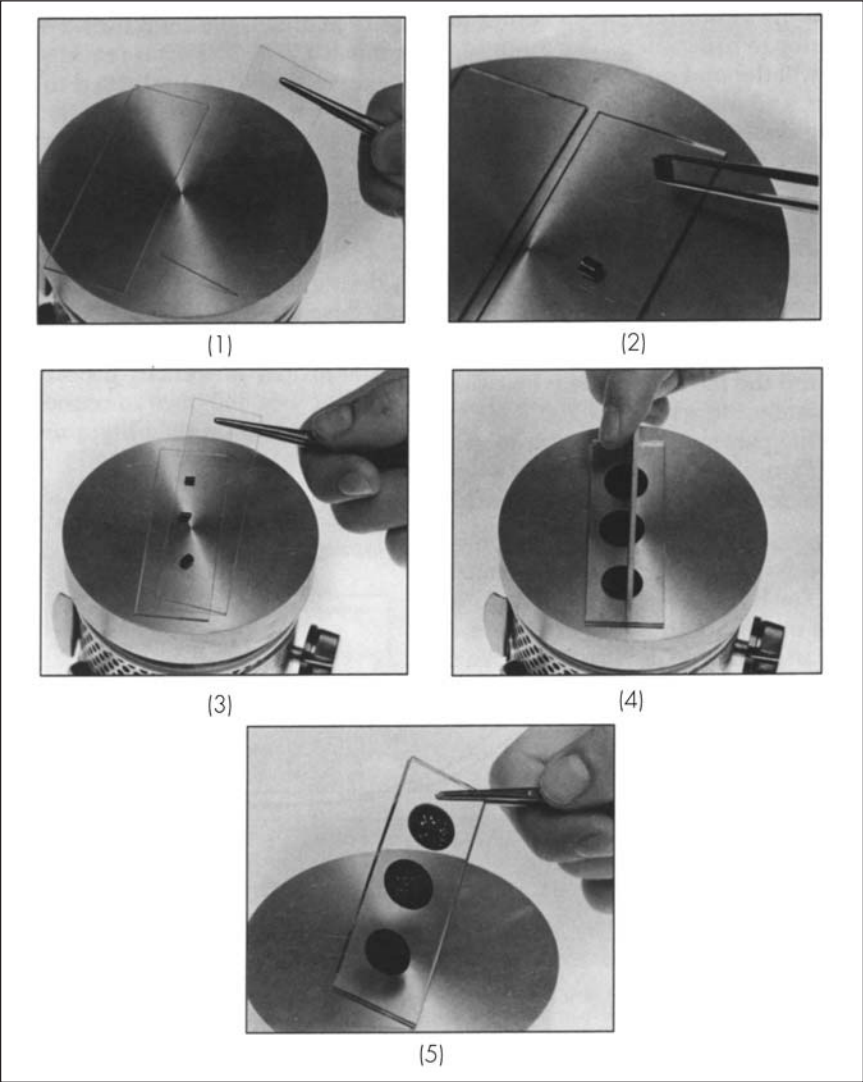


Figure 9-9a. TVI moisture content testing. (Courtesy GE)

material. These tests must be performed on specifically designed and processed samples and not on the actual product itself. As with most tests, the samples must be properly conditioned prior to being tested. These tests must be considered only for reference. Results are used to aid selection of the proper plastic material for a specific application.

Tensile Strength

Tensile strength testing (ASTM #D-638 [ISO #527-1 and 2:1993]) is performed to determine the point at which a plastic sample either breaks or yields. The measurement is useful in selecting materials that will be exposed to tensile-type (pulling) actions.

The test is performed by gripping a sized and conditioned sample in a machine with a movable head and a stationary head, as shown in Figure 9-11. The moving head is activated and the sample is slowly pulled apart. Measurements are taken at the point of breakage or yielding. These are expressed as pounds per square inch (Pascals).

Compressive Strength

Although compressive strength testing (Figure 9-12) is seldom utilized, it does aid in determining the ability of a plastic to withstand the application of compression forces. Results of the tests (ASTM #D-695 [ISO #N/A]) may be useful in designing products such as those used for supporting shelves or overhead structures. Normally, flexural strength and creep resistance testing are preferred to compression testing.

The test is similar to tensile testing, but instead of pulling the sample apart, the compression test pushes it together until it breaks or yields.

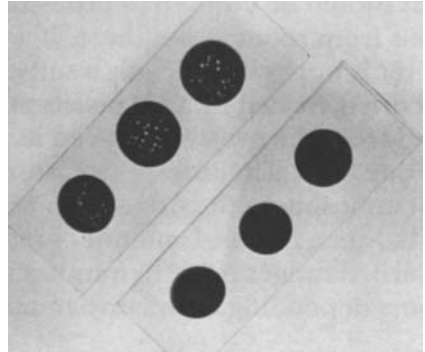


Figure 9-9b. TVI patterns on test slide. (Courtesy GE)

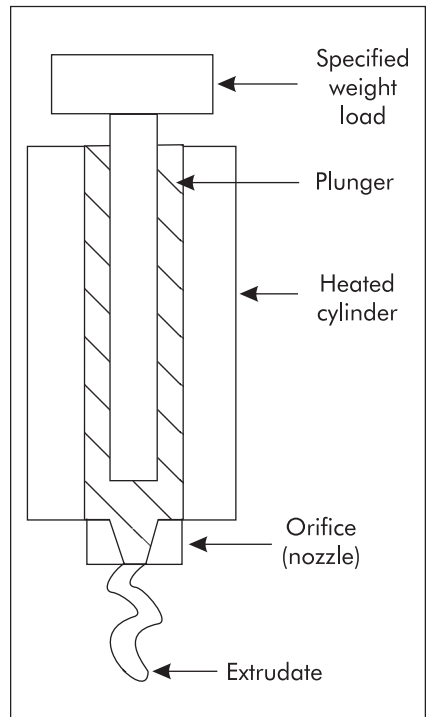


Figure 9-10. Melt flow index.

Table IX-1. Melt Index Value’s Impact on Plastic Properties

As melt index value <i>decreases</i> :	
Stiffness	Increases
Tensile strength	Increases
Yield strength	Increases
Hardness	Increases
Creep resistance	Increases
Toughness	Increases
Softening temperature	Increases
Stress-crack resistance	Increases
Chemical resistance	Increases
Molecular weight	Increases
Permeability	<i>Decreases</i>
Gloss	<i>Decreases</i>

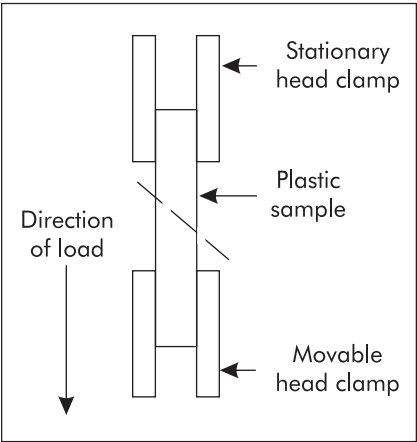


Figure 9-11. Tensile strength testing determines break and yield points in plastic materials.

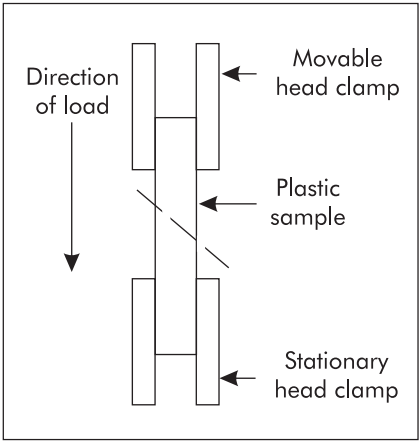


Figure 9-12. Forces in compressive strength testing push toward a stationary sample until the material yields.

Flexural Strength

Flexural tests are related to the stiffness of the plastic and its resistance to bending. The test (ASTM #D-790 [ISO #178:1993]) begins by placing a flat specimen across two beams similar to that shown in Figure 9-13. An opposing load is then applied to the specimen until it either breaks or yields.

Creep

While creep (or deformation under load) information may be valuable for designing products which will support a load over a long period of time, there is no established method of determining creep. The ASTM has provided a method that is frequently used (ASTM #D-674 [ISO #N/A]), but owing to the complications involved with measuring creep, results of this test should be used only for research and not definitive, routine material and application specification.

The test consists of clamping a specimen at one end, and hanging a load on the other end, as shown diagrammatically in Figure 9-14. The amount of stretching that takes place is measured first hourly, then daily, then weekly, and finally (if required) monthly, for a total time period of up to one year. Any measured stretching (creep) is plotted on a graph which shows the tendency and rate of creep for a specific plastic. However, again, there are so many variables that come into play with this test that use of results is limited to reference only.

Impact

Two basic test methods (ASTM #D-256 [ISO #179 and 180:1993]) are used for determining impact strength: the IZOD (vertical beam) and Charpy (horizontal beam) tests. These tests are performed on both notched and unnotched samples. Figure 9-15 shows the notched method.

For both tests, a recording is made of the travel of the impact head (pendulum) with both no specimen mounted and with specimen mounted. The difference is calculated as an energy loss by the pendulum and is referred to as the impact resistance of the plastic.

Thermal Testing

There are five basic measures of note in testing for thermal properties of plastic materials: melting point, heat deflection temperature, vicat softening temperature, flammability, and oxygen index. Tests for each of these are performed to determine end-use properties as well as processing parameters.

Melting Point

The *melting point* is referred to as the *melting temperature* (T_m) for crystalline materials and the *glass transition* (T_g) for amorphous materials. It can be thought

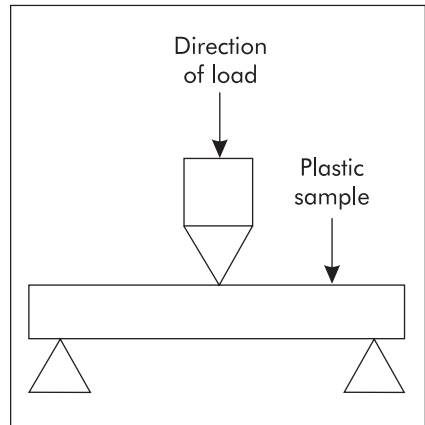


Figure 9-13. Flexural strength testing.

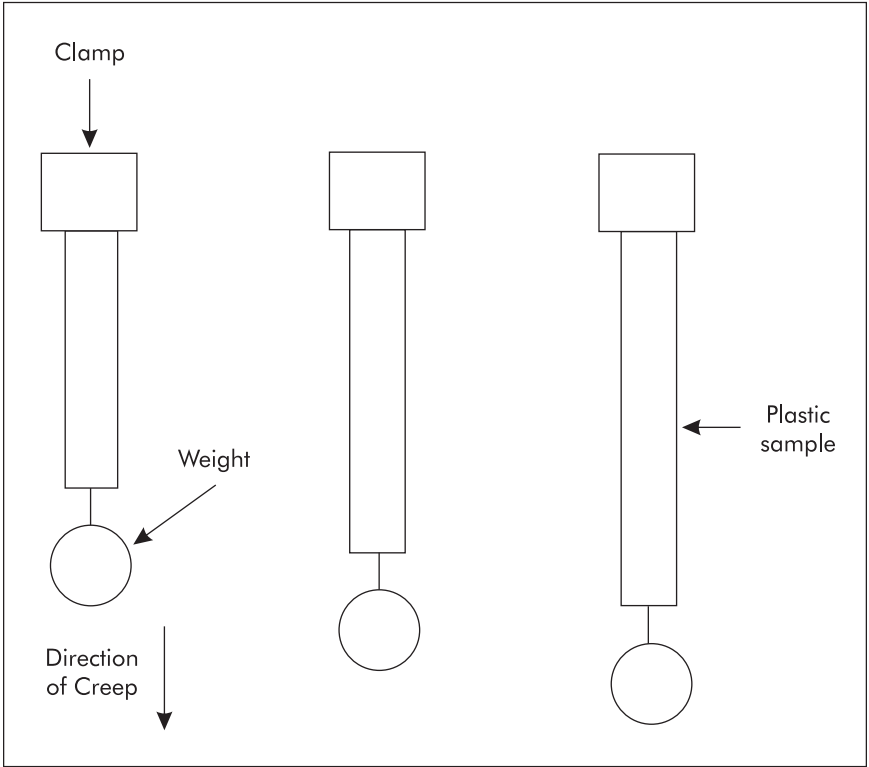


Figure 9-14. Creep testing.

of as the temperature at which the plastic material is readily flowing and able to be properly injected. It can be used to determine the starting temperatures at which the injection barrel of the machine can be set for initial processing.

The test (ASTM #D-3418 [ISO #3146:1985]) is commonly performed using a Differential Scanning Calorimeter (DSC), a machine that measures the temperature difference and energy necessary to establish a “zero” temperature difference between a specimen and a reference sample. It records this data as a curve. In Figure 9-16a this curve is dramatic and shows a definite peak. The tip of this peak is the temperature at which the crystalline material reached the melting temperature. The curve in Figure 9-16b does not have a sharp peak because the material is amorphous, not crystalline. However, by magnifying the area in which the glass transition point should be found, we will see in the inset a pattern on the line that resembles an “S.” By drawing a line through the curves of this “S” we can determine the exact temperature at which the amorphous material goes through the glass transition.

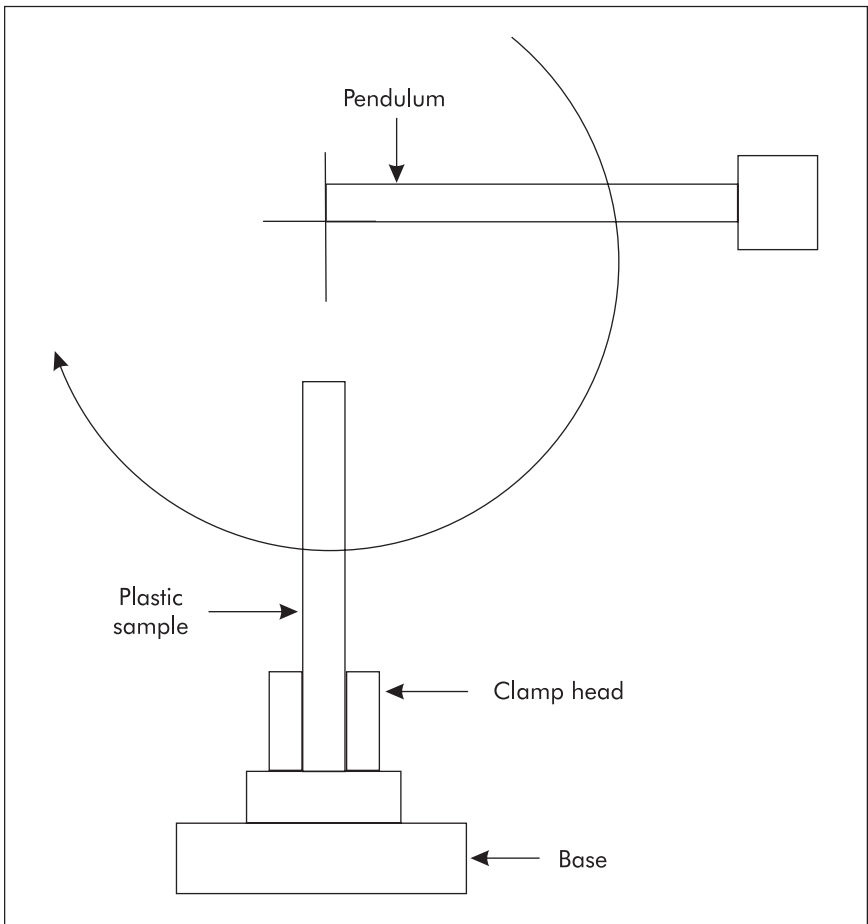


Figure 9-15. Notched impact tests.

Heat Deflection Temperature

The *heat deflection temperature* (HDT) (Figure 9-17) is a good reference point for determining the temperature at which molecular action takes place and the plastic material can flow. It should *not* be used to determine the end-use temperature limits of a specific product design. In fact, it is not a practical test for any reason because it is performed under a load and does not simulate any product design situation, unless one is designing a product to fail at a specific temperature under load. This test should be used for reference only.

The test consists of placing a specimen edgewise as a beam over two support points 4 in. (102 mm) apart. This apparatus is placed in a heated liquid bath

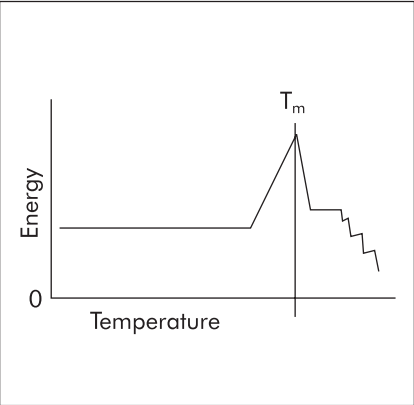


Figure 9-16a. Crystalline melting temperature (T_m) graph, exhibiting its recognizable peak.

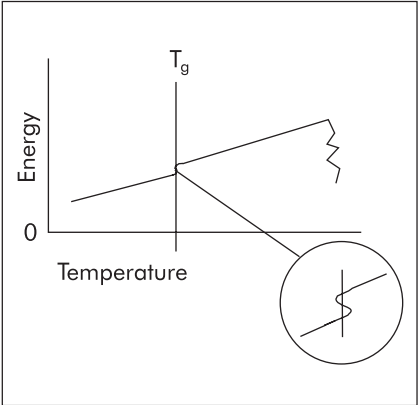


Figure 9-16b. Amorphous glass transition (T_g) has no peak, but is recognizable by its S-shaped transition point.

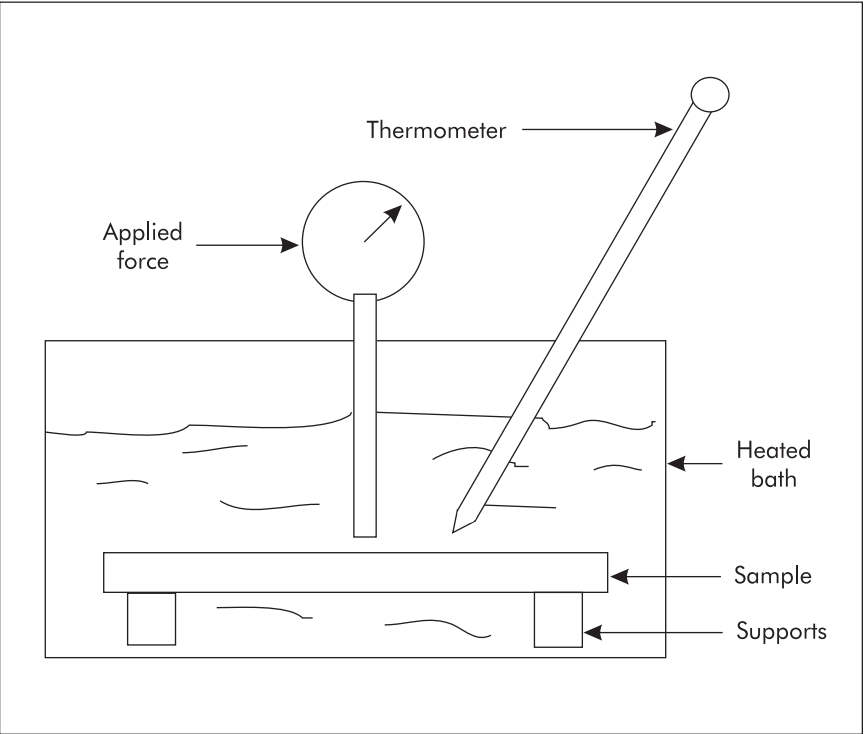


Figure 9-17. Heat deflection temperature testing.

capable of maintaining the estimated HDT for the specific plastic being tested. The temperature of this bath begins at room temperature and is increased at the rate of 36°F (2°C) per minute. A load is placed against the edge of the specimen and the temperature at which the specimen deflects to a total of 0.010 in. (0.254 mm) is recorded as the heat deflection temperature.

Vicat Softening Temperature

The Vicat softening temperature test (ASTM #D-1525 [ISO #306:1987]) is similar to the HDT test, but the specimen is not placed on edge or on support beams; it is placed flat at the base of the apparatus which is then placed in a suitable heated liquid bath.

As illustrated in Figure 9-18, a needle probe is placed against the surface of the plastic and a specific load applied to the probe. The temperature of the bath is slowly increased until the needle penetrates the plastic specimen to a total depth of 0.040 in. (1.0 mm). This is recorded as the Vicat softening temperature.

Flammability

The accepted test for flammability (ASTM #UL-94 [ISO #UL-94]), Figure 9-19, is performed under the guidelines presented by Underwriters Laboratories (UL). These

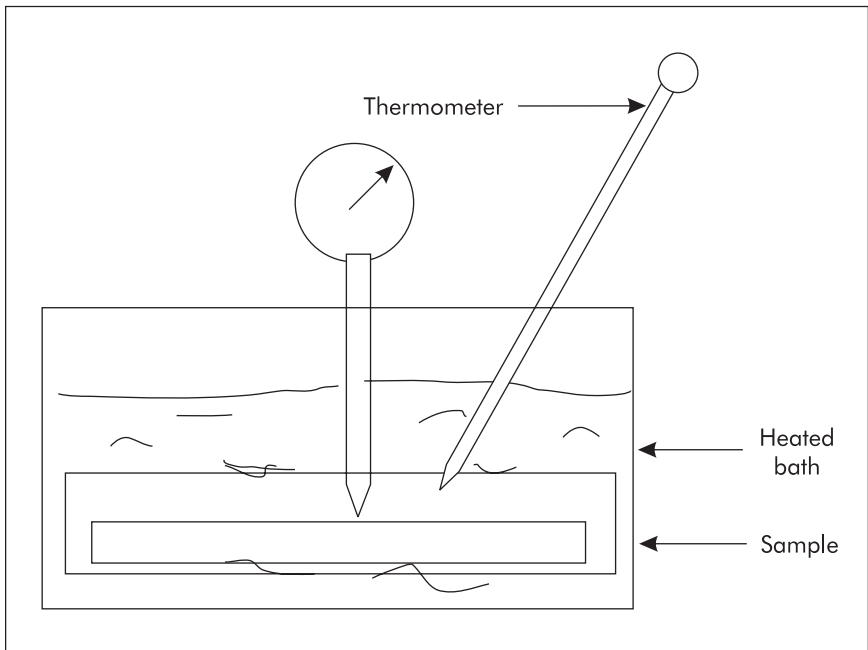


Figure 9-18. Vicat softening temperature testing.

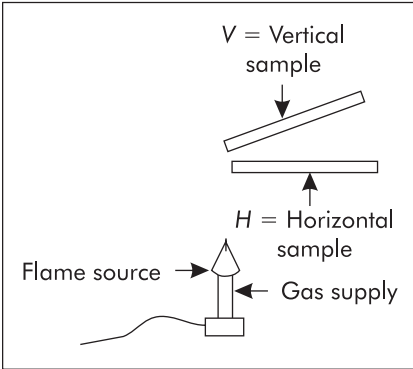


Figure 9-19. Flammability testing.

tests are performed on specimens placed either horizontally (*H*) or vertically (*V*) to a flame source, depending on the plastic being tested.

The measurements to be recorded include the condition of the plastic as an ignition source is applied, the condition when the ignition source is then removed, then again as the ignition source is reapplied. Any smoking, dripping, or other problem must be recorded, and the speed and distance of any flame travel calculated. The test must be run numerous times and numerical values stated for the various conditions noted. These are added together and the UL rating is specified as a result of the total value.

It must be noted that this test is *not* to be performed on a product but only on a specially-molded sample of the specific material being tested. Also, the rating must state a thickness at which the rating applies. This is normally 0.062 in. (1.6 mm) but other thicknesses are acceptable. Generally speaking, the lower the thickness mentioned in the rating, the more flame retardant the plastic material.

Also, the rating must state a thickness at which the rating applies. This is normally 0.062 in. (1.6 mm) but other thicknesses are acceptable. Generally speaking, the lower the thickness mentioned in the rating, the more flame retardant the plastic material.

Also, the rating must state a thickness at which the rating applies. This is normally 0.062 in. (1.6 mm) but other thicknesses are acceptable. Generally speaking, the lower the thickness mentioned in the rating, the more flame retardant the plastic material.

Limiting Oxygen Index

This test (ASTM #D-2863 [ISO #589:1984]) is being used, in many cases, to replace the UL-94 flammability test because it appears to be more applicable to molded products rather than the UL requirement of specially-molded specimens. The limiting oxygen index, or LOI, test is used to measure the minimum amount of oxygen that will support flaming combustion of a plastic product.

The test consists of mounting a specimen vertically in a tube, similar to that shown in Figure 9-20. A flow of

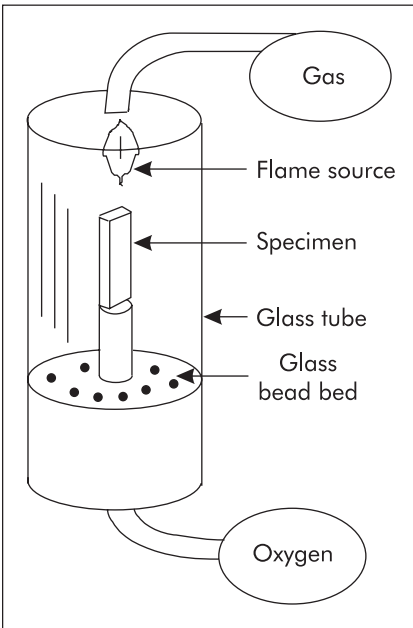


Figure 9-20. Limiting oxygen index (LOI) testing.

air is passed through the tube with a specific percentage of oxygen included. The specimen is ignited with a flame source; the source is then removed and recording is begun. The oxygen level is adjusted upward or downward to determine the minimum level that will maintain burning of the specimen for a specific amount of time. This level is stated as the percentage of oxygen contained in the air stream.

FAILURE ANALYSIS

As a Design Tool

While failure analysis (FA) commonly concerns finished products that may have failed during use, it also can be considered a method of analyzing product design and stability to determine causes of defective (but not failed) parts, as well as causes of defective processes related to molding and finishing that part. Failure analysis tools can be used to predict what might happen if certain actions are taken regarding design changes, tool changes, material changes, or process changes. Failure analysis differs from troubleshooting in that FA is usually performed at some point after the product is molded, while troubleshooting is usually performed during, or *immediately* after, the molding process.

Failure analysis is a concept rather than a method. It may require sophisticated instruments and tools in some cases, but in most cases visual observation is all that is necessary. An example of this would be the analysis of surface cracks resulting from normal use of a flat plaque. Visual observation may ascertain that one corner of the plaque was being held up in the mold during ejection. This would cause a stress condition in that corner which might not be visible during normal inspection procedures. It would, however, be emphasized if the part was ever physically struck on that corner, releasing the stress, resulting in cracks.

FA can be thought of as a form of reverse engineering. A failed part is inspected, an initial determination is made as to what analysis equipment, if any, is required, and samples are gathered or created. In addition, as much data is recorded as possible regarding conditions existing at the time of the failure. This may require a separate investigation.

Stress

Most failures of plastic products can be traced to stress. For our purposes, stress can be defined as *resistance to deformation from an applied force*. All this means is that the plastic molecules (either molten or solidified) are trying to travel one way and something is trying to force them to travel a different way. This results in stress in many forms. The five most common forms found in injection-molded products are tension, compression, bending, twisting, and shear.

These forms of stress can be created by:

- Moisture in the material;
- Improper temperature profile in the heating cylinder;
- Hot and cold spots in the mold (more than 10° F [5.6° C] difference between any two points);
- Half of the mold hotter than the other;
- Gating into a thin section rather than a thick section of the part;
- Improper injection pressure profile;
- Inconsistent cycles;
- Material degradation;
- Excessive regrind usage;
- Physical obstructions in the mold, such as burrs on metal edges;
- Improper runner design; and
- Many other situations.

It should be obvious that if so many things can cause stress, stress is probably going to occur on a regular basis. And when stress is present, there will be product failures. Stress conditions must be minimized as much as possible. But, with all of the awareness and minimizing efforts, failures due to stress and other situations may still occur. In the remainder of this chapter we look at some of the methods and equipment used for determining the causes of those failures.

Differential Scanning Calorimeter

The Differential Scanning Calorimeter (DSC) (Figure 9-21) is a device that compares the amount of energy (in calories) that is required to establish a zero temperature difference between a substance (plastic) and a reference specimen. The DSC generates a curve which can be plotted and analyzed.

As with other testing, sample preparation is key to successful DSC testing. The sample size is normally less than 0.006 oz (17 mg), and samples can be prepared either from raw material (regrind or virgin), or from the molded product itself.

Stress

If stress is present in a molded part, it will be defined as a “spike” on the DSC curve. Figure 9-22a shows this spike as it appears on the curve for a crystalline material. It will also appear on the curve for an amorphous material, but it is not as evident.

Moisture

If moisture is present in a raw material, either virgin or regrind, it, too, will show up on the DSC curve as a spike, but the spike will be located at the 212° F (100° C) mark, where the moisture turns to steam (see Figure 9-22b).

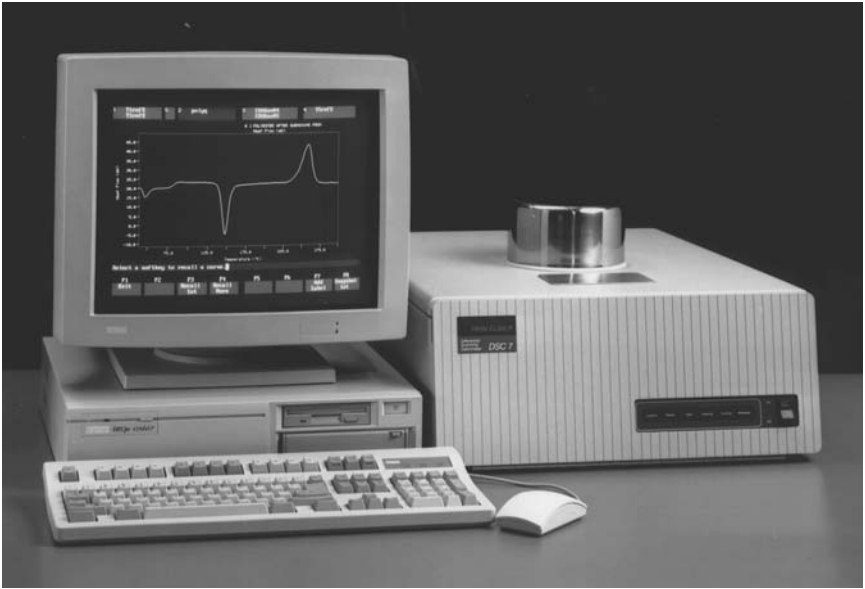


Figure 9-21. Differential Scanning Calorimeter (DSC). (Courtesy Perkin-Elmer Corp.)

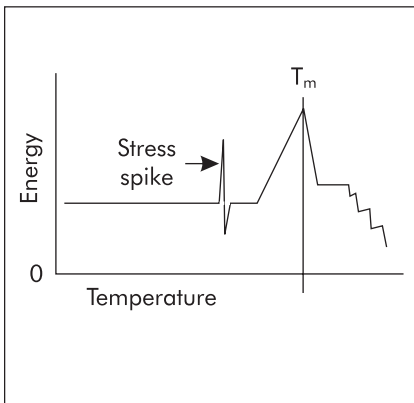


Figure 9-22a. Stress spike on DSC curve for a crystalline plastic.

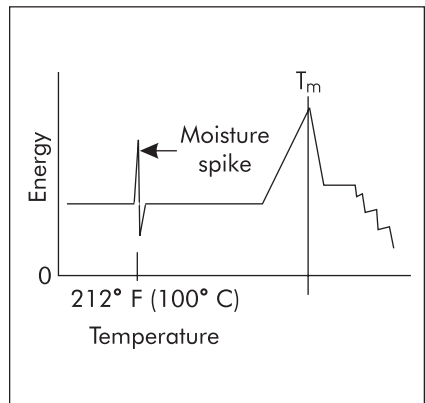


Figure 9-22b. Moisture spike on DSC curve.

T_m and T_g Points

The DSC curve displays the melting temperature (T_m) of crystalline materials and the glass transition (T_g) temperature of amorphous materials. See Figures 9-16a and b for examples of these DSC curves.

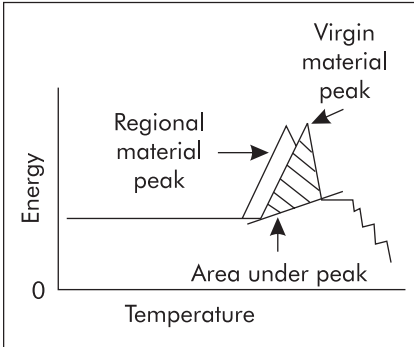


Figure 9-23. Determining regrind percentage (crystalline material).

Regrind Percentage

Excessive regrind can cause degradation and brittleness in the molded product. The product can be analyzed by running a DSC curve, which will display two peaks (crystalline materials) or two transition points (amorphous).

In Figure 9-23, the curve shows the two DSC peaks. The one on the right is for virgin and the background one on the left is for regrind. The reason there are two peaks is that regrind melts at a lower temperature than virgin. The DSC will calculate the area beneath the two

peaks. Regrind percentage is calculated by establishing the percentage of the total area that is comprised of the regrind area.

Crystallinity

The crystal structure of a crystalline material breaks down when exposed to the heat of the injection barrel. For the molded product to attain maximum structural properties, this crystal structure must be allowed to re-form as the material cools and solidifies. If the material cools too quickly, only a portion of the crystals is allowed to re-form and structural integrity suffers. The usual amount of recrystallizing that is acceptable is 85%. The DSC will determine crystallinity percentage through a comparison of two samples, the first a known, fully crystallized specimen, the second a specimen taken from the part being tested (see Figure 9-24).

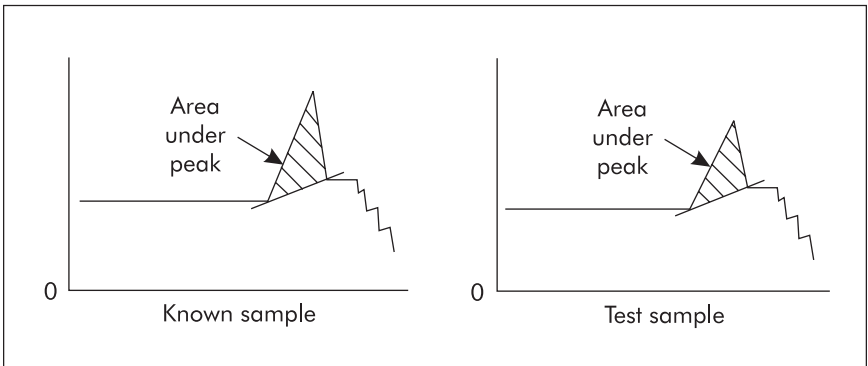


Figure 9-24. Crystallinity curves.

To calculate crystallinity, the area beneath the peak of the sample being tested is compared to the area beneath the peak of the known sample. The ratio that develops converts to the percentage of crystallinity that was attained when the tested sample was originally molded. This method is used to determine the degree of “cure” of thermoset materials.

Calculating Glass Content

Furnace Method

A relatively simple test, the furnace method of determining glass content is performed by first weighing the molded sample. The sample is then placed in a high-temperature muffle furnace (Figure 9-25), capable of maintaining approximately 1000° F (538° C) for extended periods of time.

After 30 minutes to 2 hours, the sample is removed, allowed to cool to room temperature, and weighed again. The difference between the first and second measurements is the percentage of resin, binder, and fillers that was burned away. The remaining material represents the percentage of glass reinforcement that is present. This test also can be performed using a hand-held gas torch in place of the furnace.

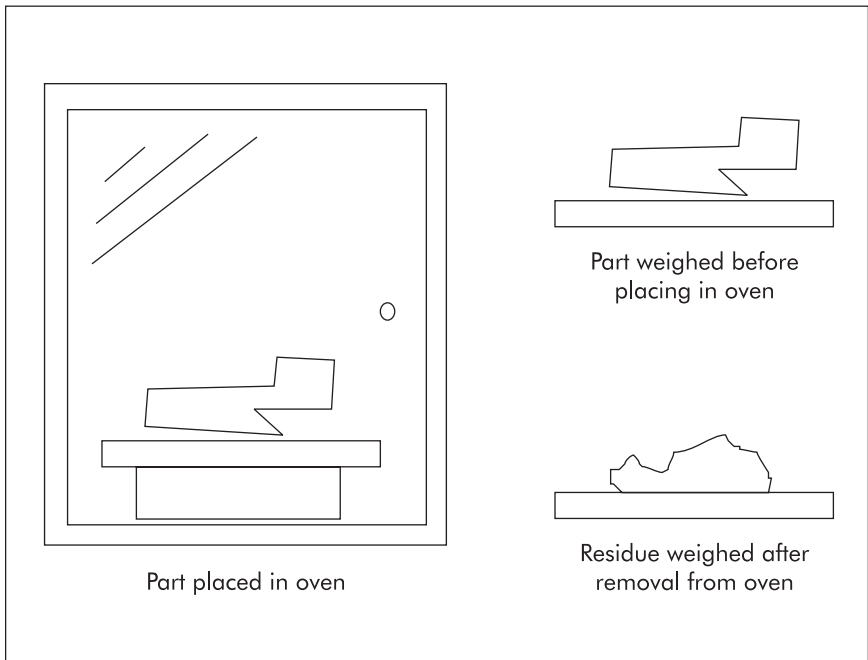


Figure 9-25. Samples in furnace.

TGA Method

Thermogravimetric analysis (TGA) is a process that can be used to determine filler content, resin content, and content of other components of a molded product or raw material.

The process consists of placing a small specimen in a chamber that is part of the TGA apparatus (similar to the one shown in Figure 9-26a). This chamber continuously weighs the sample and creates a curve displaying that weight. The chamber is slowly heated to approximately 1000° F (538° C). As the sample heats, the components in the plastic are burned. The curve shows a peak at each point this happens until there is nothing left but ash. This ash is the reinforcement material (fiberglass) that was used in the plastic. The curve generated (Figure 9-26b) shows the weight of residue left in the chamber at each peak, so a percentage can be calculated from that data.



Figure 9-26a. TGA chamber. (Courtesy Perkin-Elmer Corp.)

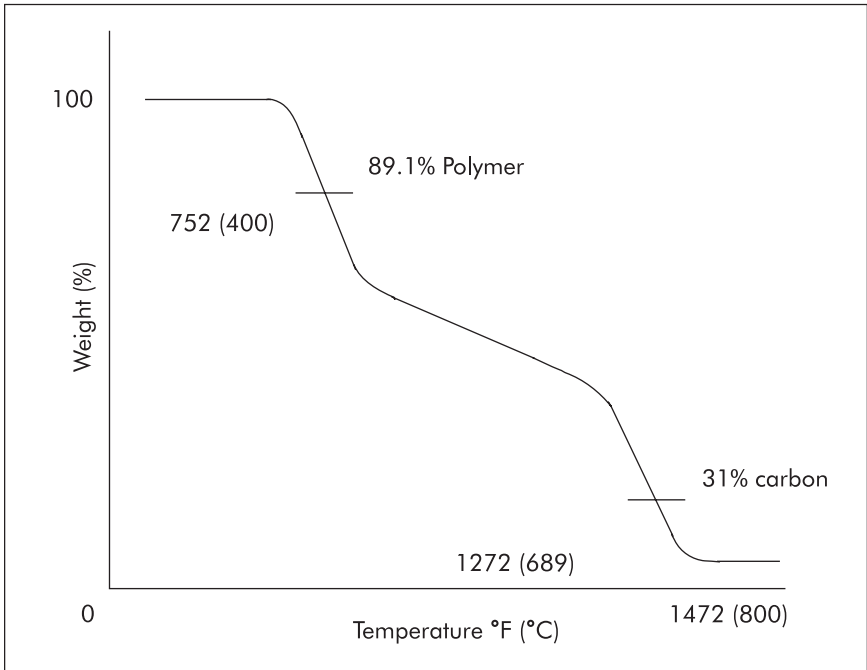


Figure 9-26b. TGA curve. (Courtesy Perkin-Elmer Corp.)

SUMMARY

Proper conditioning of the samples is the most important item to remember when performing tests on plastic parts or materials.

The five most common electrical tests are dielectric strength, dielectric constant, volume resistivity, surface resistivity, and arc resistance. The five most common physical tests are shrinkage rate, density, water absorption, moisture content, and melt flow index. The five most common mechanical tests are tensile strength, compressive strength, flexural strength, creep, and impact resistance. The five most common thermal tests are melting point, heat deflection temperature, Vicat softening temperature, flammability, and limiting oxygen index.

Failure analysis differs from troubleshooting in that FA is usually performed after the product is molded, while troubleshooting is usually performed during the molding process. Stress, the number-one cause of product failure, is a resistance to deformation from an applied force and, as such, is the result of molecules being forced to travel in directions and methods which oppose their natural tendencies. A Differential Scanning Calorimeter (DSC) indicates the presence of stress in a molded part.

The DSC also can be used to test for the presence of moisture in raw material, the percentage of regrind present in a molded part or raw material mix, and the degree of crystallinity present in a molded product.

The percentage of glass reinforcement available in a molded part or raw material can be determined by use of either the furnace method or the TGA method. Both methods consist of burning away all the organic material from the plastic and leaving the glass.

QUESTIONS

1. What are the two primary sources of information regarding test procedures for the plastics industry?
2. What will improper sample preparation cause?
3. Name three of the five common electrical tests.
4. Name three of the five common physical tests.
5. Name three of the five common mechanical tests.
6. Name three of the five common thermal tests.
7. State the main difference between failure analysis and troubleshooting.
8. What is the definition of *stress* as defined in this book?
9. What is the full name of the test equipment known as DSC?
10. How is regrind percentage determined using the DSC?
11. How is crystallinity percentage determined using the DSC?
12. Name the two common methods for determining percentage of glass content.

The Product Design Session

10

BRAINSTORMING

It is an exciting experience to be involved in the evolution of a product, from design concept to finished part. The first steps are cautious, but soon the adrenaline is flowing and an idea begins to be shaped into a visual, then actual, product. During the early stages of development it is common for an individual to remain quiet and protective of an idea or concept. But, at some point, that individual finds a need to express the idea, or to solicit opinions and other ideas from as many sources as possible. This can take the form of a library search to see if similar products already exist, or it can take place as a brainstorming session in which the designer presents a concept and others join in to give constructive criticism and positive feedback. This type of session is usually regarded as extremely beneficial for both the designer and the attendees. They all bring valuable information and expertise to the session and all take away a new and fuller understanding of the capabilities and level of expertise of each other, as well as the excitement of the possibility of launching a new product concept for their company.

Who Are the Players?

The initial brainstorming session need not be more than an hour in length. This is not the time to optimize every aspect of the product design or manufacturing process, but a time for introduction of the primary design concept and awareness of what prompted the idea. It is intended to point out both the good and bad aspects of a design or concept, and to find ways of improving the design, or simply to establish what needs to be done to bring the product to life. While it is not wise to have too many people at any one meeting, at a minimum the attendees should include individuals representing the Product Design team, Purchasing, Materials Engineering, Manufacturing Engineering, Quality, and Marketing. It is always possible that two or more groups could be represented by the same individual. For example, Manufacturing Engineering and Materials Engineering may actually be the same department. For this most important first meeting, each group should have no more than one representative in attendance, with future attendance determined by specific requirements. For instance, while

Manufacturing Engineering may be represented by a person with expertise in plant layout, the next meeting may require someone with tooling expertise, so a different individual might attend. Or the most time-effective method might be to utilize the same representative at every meeting, relying on that individual to be the focal point for relaying information in both directions, from the meeting and to the meeting. It would be dependent upon that person to solicit the necessary assistance and provide expertise and information to the brainstorming team, regardless of where that data came from. This keeps the majority of time-consuming activities outside the main meeting, yet in the hands of the people best suited to provide the necessary expertise. It also keeps the attendance at the main meeting to a minimum, which minimizes lengthy discussions, and is beneficial in setting up the meetings to accommodate the schedules of the attendees.

What Takes Place?

The primary reason for this first meeting is simply to acquaint the attendees with the initial design concept, solicit opinions, and introduce the players. Letting the players get to know each other may seem elementary, but in larger companies, or companies with a multitude of sites, it is quite possible that some of the individuals have never met each other. This is the time for those introductions to be made. The meeting should have been arranged by a facilitator, one who does not actually contribute to the technical part of the meeting, but instead serves as monitor of the proceedings. The main function of the facilitator is to keep the meeting in order and on time. The facilitator has sole discretion as to the length of each person's discussion time and also keeps the discussion focused on the subject at hand. He or she has the responsibility of arranging the meeting schedules, keeping all attendees aware of upcoming meetings, and reminding representatives of any commitments they may have made. This person also has responsibility for formally contacting the manager of each of the proposed attendees to solicit their support, who in turn should arrange for the proper individual to attend the meeting. Protocol presumes that the attendance of the individual signifies the support of the manager.

Following introductions, the facilitator turns the floor over to a representative of the Product Design department who presents the scope of the design concept. During this presentation, the attendees are expected to consider the part their department or function can play in bringing the concept to reality. When the presentation is completed, the floor is opened for questions from the representatives. If none are forthcoming, the facilitator has the responsibility of asking leading questions, such as "what is estimated annual usage?" or "what materials have been selected for production?" These are intended to start a dialog between all attendees. If this is not successful, the moderator may elect to adjourn the meeting, informing all attendees to return to the next meeting prepared to discuss their roles. Or, in lieu of questions (or in addition to), the

facilitator may simply start things off by asking each attendee to relate their perception of how they can assist in bringing the project to fruition.

It is vitally important to establish a timetable at this first meeting. This can be done ahead of time by the design team, on its own, by way of contacting individuals such as manufacturing engineers and purchasing personnel who can contribute to the determination. They can estimate tooling times and other time line concerns and bring those estimates to the meeting. Or time can be set aside at the meeting to discuss all of the elements needed to make the time line estimates. In either case, a schedule must be set. This schedule should be such that each phase of the project is identified and given a range of time for completion. Minute details do not have to be considered at this point, just a “stake in the ground” to get the project rolling and a target date by which production needs to start. Once this schedule is established, the meeting can be adjourned and all attendees are expected to determine their roles in meeting the schedule dates.

During this meeting also, the design team should ask for assistance in ensuring proper material selection, establishing the number of cavities for the molds, selecting the proper manufacturing process, and any other help that would assist in making a product of the highest quality, at the most effective cost, in a timely manner, and at the projected annual volumes. This is the time to gather such information if it was not already obtained during the initial design process. The design team should exhibit an ongoing willingness to welcome the advice of other attendees.

There are a variety of questions and concerns that should be answered during the brainstorming meeting, ranging from describing the product shape to explaining how much it should cost. These are examined in the following paragraphs.

What Does the Product Do?

At the meeting, the design team has the opportunity to describe the primary (and secondary, if appropriate) function of the new product. This should include how it works as well as what it does. Sketches or full drawings, if possible, should either be made available to each attendee or exhibited at a central location for all to view. Open discussion should be encouraged after the initial presentation is made. This should result in recommendations from the attendees that may help create new aspects for the design, allowing it to perform its functions better or perform additional functions that will enhance its marketability.

At this session it is advisable to explain why the product is needed. Does it perform a function that was not available before? Or does it perform a function better than an existing product? Possibly, it is simply designed to be a less expensive version of something that already exists. Or it may be a product that takes advantage of a new technology. There are a multitude of reasons for designing a product. The appropriate ones should be explained to the attendees so

they can understand the reason they have been asked to participate in bringing the product to the marketplace.

How Will the Product be Made?

Once everyone has had a chance to see the drawings, sketches, or models that have been presented, it is time to consider what methods will be used to manufacture the new product. For our purposes here, and exercises that follow, we are assuming that injection molding is the method of choice. We can assume that this came about as the result of attendees making that decision based on quantities, basic design features of the product, the desire to minimize or eliminate secondary operations by incorporating bosses, holes, attachment points, snap fittings, etc., into the product design, and the material selected. We address these decisions later.

Once the decision of how to manufacture has been made, a discussion should take place (either at the meeting, or in a subsequent meeting) as to what size equipment and tooling is required to meet the projected annual volumes of the product. It should be understood that the first year's requirements will be lower than subsequent years, and that there will be a "ramping-up" of production capacity during the first year. Ramping-up refers to the practice of starting production at a low, limited number, and slowly building up capacity to meet demands. This is more practical than trying to start production at the full capacity of equipment and tooling that will be required for maximum production. The normal ramp-up procedure consists of producing approximately 10% of the expected annual volume during the first quarter of actual production. This is increased to approximately 30% during the next quarter, and 50% during the third quarter. And, finally, during the final quarter, production is established at a full 100% of projected volumes. Obviously, these numbers can be adjusted to suit specific applications. For instance, if the expected life cycle of a product is only 2 years, it may necessitate reaching the 100% figure during the second quarter, instead of jeopardizing market share by waiting until the final quarter.

The information compiled during this phase of the meeting can be taken back by the respective groups to be used in making the final determinations regarding manufacturing methods and the equipment and tooling required. It is understood that final numbers and estimates will be presented at the first opportunity to the rest of the attendees at a future meeting.

When Should Production Start?

Commencement of production is the most important time detail required for determining all other activities of the production program. It is usually based on information gathered by Marketing for which they have documented probable customer response to the design concept. This date is not simply a target that someone guessed at, although *educated* guesses may play a role in determining

the date. Usually, the date is negotiated by the members of the brainstorming team. This group may have evolved into a full-fledged production program group at this point, which usually happens quickly after assembling the first brainstorming members. The members all gage the level of their respective activity and estimate the time required to perform those activities. Normally, this time is far too long to meet the original date conceived by Marketing, and negotiations ensue. It should be stated that Marketing has a very strong influence on the production start date because they have their collective fingers on the pulse of the end users across the marketing area. While this may be argued by any other member of the program team, it does not alter the fact that Marketing will probably have final say as to what the production start date will be. It is up to the other members to find a way to meet that date. The approach should *not* be one of “standing ground” to fight off any attempt to get one member or the other to commit to an unreasonable date. The approach should instead be one of pulling together to find the ways and means of achieving the production goal in the time allotted. It should be looked at as a challenge, but one which can and should be successfully met.

Where Will It be Built?

It may be determined that this product cannot be made (at least entirely) in house. Allotted time may be too short to accommodate resources available to in-house personnel, or equipment requirements may be outside the available limitations. Or it may be that the level of internal expertise does not match that required for the project. Moreover, it may be that there is not enough space available to properly accommodate the program. Finally, it may be apparent that internal manufacturing costs are too high, making it prohibitive to attempt to produce the parts in house. These are all valid determinations and should not be set aside simply because “it would be nice to do it ourselves.” If it makes economic sense to have the product manufactured by outside vendors, then do so. Every facet of the program can be looked at from an economic viewpoint. Even quality can be approached economically from the standpoint of what it may cost to perform the quality activities internally versus externally. The only time a decision should be made against an obvious economically-sound external production would be in the case of security. This can be a major issue in an industry with short product lifetimes, or one in which technology advances must be held in a confidential state as long as possible. And even this scenario can have an economic price tag attached to it. So the decision to make it in house or outside should always be an economic one.

Whether a decision is made for manufacturing the product entirely in house, or only in part, it will require additional planning to ensure that proper equipment is available or can be purchased in time to meet the schedules. Also, tooling design and construction must be started. Again, this can be done in house if

equipment and expertise is available, but also can be scheduled to go outside if necessary. Auxiliary equipment must be considered, as well as material handling equipment and space for incoming raw materials and inventory for shipment. And don't forget shipping charges. These are often overlooked in the early stages of the program and can come back to haunt you when it's time to finalize production numbers. Plastic parts are normally light in weight and shipping companies often penalize for that condition.

How Much Should It Cost?

In Chapter 7 we discussed in detail some of the variables that go into determining the manufacturing costs of an injection-molded product. At this stage of the project an estimate can be made of the cost to mold, inspect, perform secondary operations (if any), package, and ship the product. Figure 10-1 shows a typical breakdown of costs incurred for injection molding a product. It does not include secondary operations.

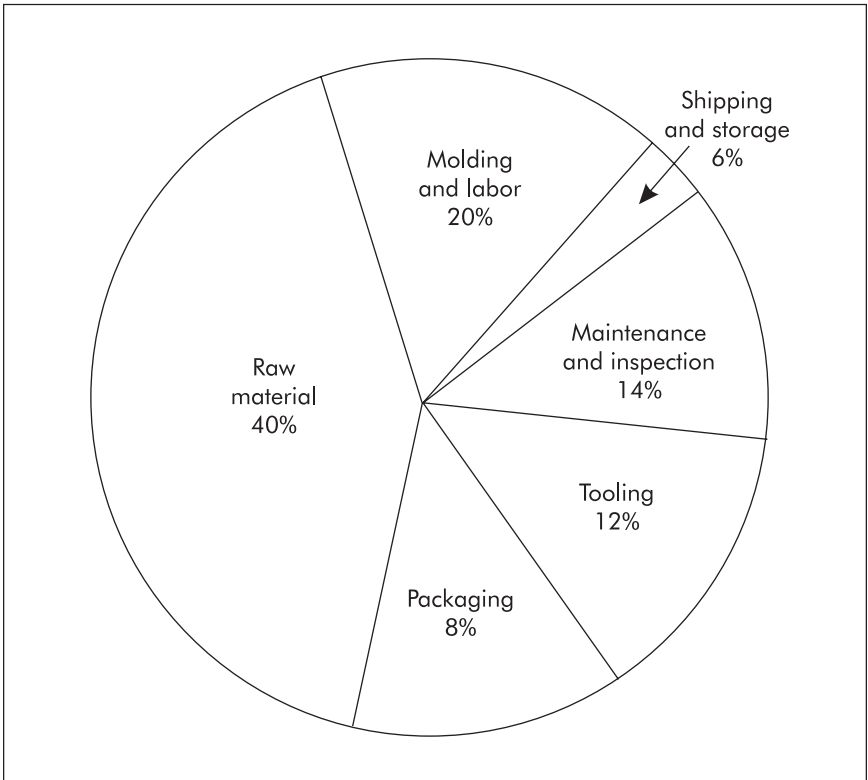


Figure 10-1. Product cost breakdown.

As the chart depicts, material is usually the largest factor in determining manufacturing costs of a molded product. In the case shown, tooling costs have been amortized over the life of the product, in this case 500,000 pieces. Maintenance and inspection costs are low and normally are about equal, so they have been combined for chart simplification.

Of course, if we are using the product for in-house applications, there would be no shipping charges, but the other charges will still exist, and we need to add storage charges. Also, at this time, we may wish to go outside for vendor estimates so we can make the “make-or-buy” decision. Many variables influence the disposition of that decision, but it all comes down to crunching the numbers and deciding whether or not it makes economic, as well as security-focused, sense to manufacture in house.

When determining manufacturing costs, we must remember storage. Whether the parts are shipped outside or used inside, they must be stored somewhere before they are used. Storage is expensive and should be minimized as much as possible. That is what created the JIT (Just-in-Time) manufacturing concept. Under JIT, parts are molded such that they are used immediately (or as close to immediately as possible) after molding. This is difficult to achieve in reality and must be considered only if very close monitoring and control of the molding process and tracking of parts are possible. Without proper equipment, controls, and computer systems, JIT manufacturing can be a disaster. With them, it can be a wonder to behold.

Project Chart Samples

The following tables are some examples of time line and project charts common in the industry. They are included here to visually describe the activities that normally take place in bringing a product from concept to fruition. It matters little what the specific product is; the charts are versatile. We are concerned with concepts here, and the charts display some of the more common ones.

Product Development

The product development chart is used as a first-pass guide to management to show the estimated time required to achieve a series of checkpoints along the path of development towards final availability of the product to the end user (customer).

We will follow the development of a product through each phase and checkpoint of Figure 10-2 as during a typical project development. Each checkpoint must be identified with an expected result, and this must be reconciled before the next phase can be completed. Each team member must sign off (approve and agree) before the checkpoint can be considered reconciled.

Phase 0—initial proposals. This is the time at which the product is proposed to team members, including representatives from Development, Marketing,

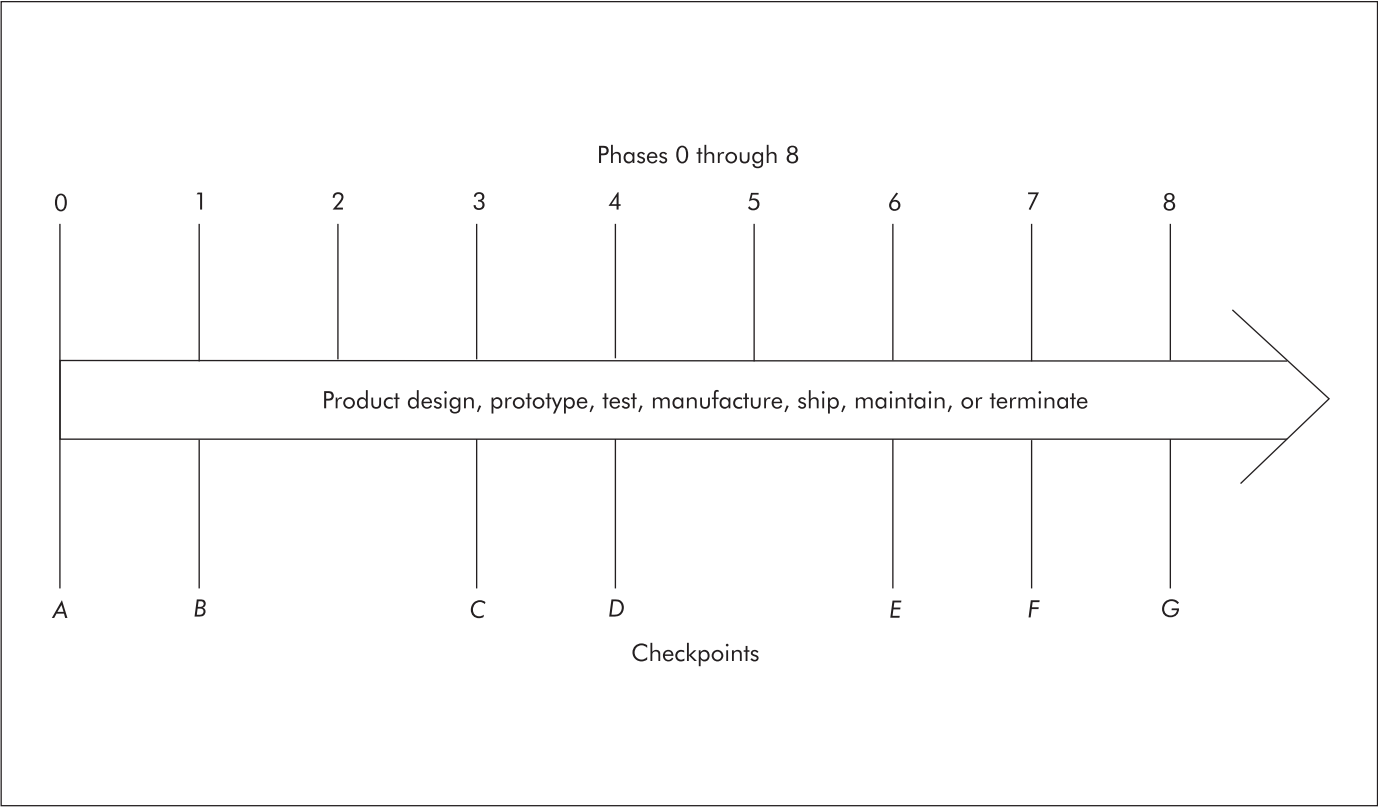


Figure 10-2. Product development chart.

Manufacturing, and Service, if applicable. These members join forces to define potential worldwide requirements and product objectives, determine if the product is feasible from a user's standpoint, and project a formal business case to solicit funding. Checkpoint A becomes a requirement to agree to the product's feasibility and purpose. During this phase, tentative dates are established for the subsequent activities.

Phase 1—define product specifications. In Phase 1 initial product specifications are prepared based on the Phase 0 proposal and succeeding discussions. In addition, initial plans are established for every aspect of the product, including target prices, schedule dates, product information, test plans, manufacturing processes, material selection, packaging requirements, etc. Funding is established at this time, and initial financial resources are committed. Checkpoint B becomes a commitment from all team members to the development plan, as it exists at this time.

Phase 2—design the product. Product design takes shape in this phase. The design team must establish initial dimensions; formalize product requirements; determine pertinent documents and notes that must be referred to; establish product form, fit, and function; and work with Manufacturing and Purchasing to select the plastic material to be utilized. This should require the input of a materials engineer, or at minimum, various materials suppliers. Typically, there is no checkpoint required during this phase.

Phase 3—design reviews. During this phase the team members go over the evolving product design to determine what changes, if any, are required. Some of these sessions can take place outside the formal Phase 3 meeting, but information accumulated must be presented to all team members. During the actual Phase 3 meeting, discussions are held and decisions made based on information available at that time. In addition, all plans are formalized, updated, and approved. Final funding requirements are established and committed. Checkpoint C becomes one of committing the product to Marketing.

Phase 4—determine sourcing strategy. During this phase determinations must be made as to where the product will be manufactured. This might require a series of considerations, such as: Can the product be made in house? Should part of it be made in house and part outside? Should secondaries (if required) be performed in house or outsourced? Where should tooling be made? Should the product be drop-shipped if made outside? etc. When the decisions have been made, formal requests for quotation (RFQ) should be created and sent to those sources capable of producing the product. While this may seem obvious if using outsources, it also should be done when using in-house facilities. It is good practice to have the in-house facility quote manufacturing packages just like the outside sources so that in-house costs can be compared with those of external sources. The RFQ should only be sent if you actually plan on producing the product. This is a legal requirement and ensures that you are not simply testing

the waters. If you merely want to know what the approximate costs will be you can ask for a “budgetary” estimate, rather than a formal quotation. This informs the vendor that you are only comparing numbers now and may not be able to proceed, and it does not bind the vendor or the requester in any way. The vendor, in turn, will not devote the normal amount of time to estimating costs. Checkpoint *D* states that all team members agree to the sourcing strategy and all quote packages are created and sent out.

Phase 5—develop prototypes. At this time, prototypes must be created. These can be in the form of computer-generated drawings and analysis programs, actual hard-copy products, or combinations of both. The prototyping strategy must be determined to establish which method should be used to create the prototypes. If only a few are needed for “warm fuzzy” feelings about the product, perhaps they can be fabricated using stereolithography or similar rapid prototyping processes. But if many prototypes are needed, or if they must be used for testing purposes, it may be better to actually mold them in tooling designed for that purpose. There are many ways of producing prototypes, and the final decision is based on specific needs. If there is a high level of confidence in the basic product design concept, it may be possible to minimize the prototype requirements and make this a parallel effort with Phase 6. No checkpoint is required at this phase.

Phase 6—validate the product. Phase 6 is used to determine the viability of proceeding into production with the product. This could be considered the last chance to drop the project, if necessary. Final costs are established, business measurements are updated, and team members must agree that the product can be manufactured to a defined quality level at the projected costs and volumes. Checkpoint *E* stipulates the approval by all team members that the product can now be formally announced and that actual production processes are validated and manufacturing can commence.

Phase 7—begin production. At this point all team members have agreed to start production, the sources have been established, material has been ordered, and production initiated. This requires proper documentation for processes, training, and quality programs, and tooling must be on its way or already in place. Marketing should have begun its work during the last phase (6), and Manufacturing should be ready to start making parts. Checkpoint *F* states that all team members agree to turn over responsibility of the project to Manufacturing.

Phase 8—terminate the program. At some point it will become necessary to terminate the program, whether for reasons of sales dropping drastically, inability to meet competitive forces, obsolescence of technology, or the product has run its planned life cycle. When it is determined to cease production, this phase of the project should be implemented to make it a formal decision. Checkpoint *G* simply becomes a requirement for all team members to approve project termination.

Project Management Process

Project management involves a multitude of duties designed to keep a variety of project activities on target toward completion. It begins shortly after a project has been conceived and continues until it has been completed. In the case of bringing a plastic product to a production level, a large number of individual projects may be involved. As seen in Figure 10-2, each phase of a program consists of many diverse projects. Each of these must be managed, either directly or indirectly, by a project manager. Some projects can run parallel with each other and be handled by the same person, but all projects must be managed. A typical flow chart of activity involved with project management might look like that shown in Figure 10-3. Project management tasks follow specific steps.

1. The first step is to identify the needs that drive the project in the first place.
2. After the needs are identified, a set of objectives can be determined and some estimates can be made regarding time, funding, and any other resources needed to complete the project.
3. Next, a risk assessment should be performed to determine whether or not the project should continue. Determinations would be based on the estimated costs and resources required to complete the project, and the availability of such resources. Also, time estimates might suggest that the project would be obsolete by completion.
4. Assuming the risk assessment determines continuation, the next step is to formalize a plan for the project by laying out all the objectives, placing time lines on each activity, beginning documentation where it is required, and establishing responsibilities for project activities and completion.
5. When the project is properly planned, the management of it must be initiated. This includes tracking progress, reporting status, conducting reviews, and controlling changes that may occur. The results of these activities are fed back through the system to become part of the project plan revisions.

Completion of these activities signals completion of the total project and final documents can be updated. This completes the project management program.

Process Management

Process management can be compared to project management in that they both require managing a series of activities that are needed to complete a program. But process management involves a different set of criteria and a different mindset. In process management, we are concerned with developing a set of phases requiring individual sets of activities to be performed. Figure 10-4 defines these activities and phases.

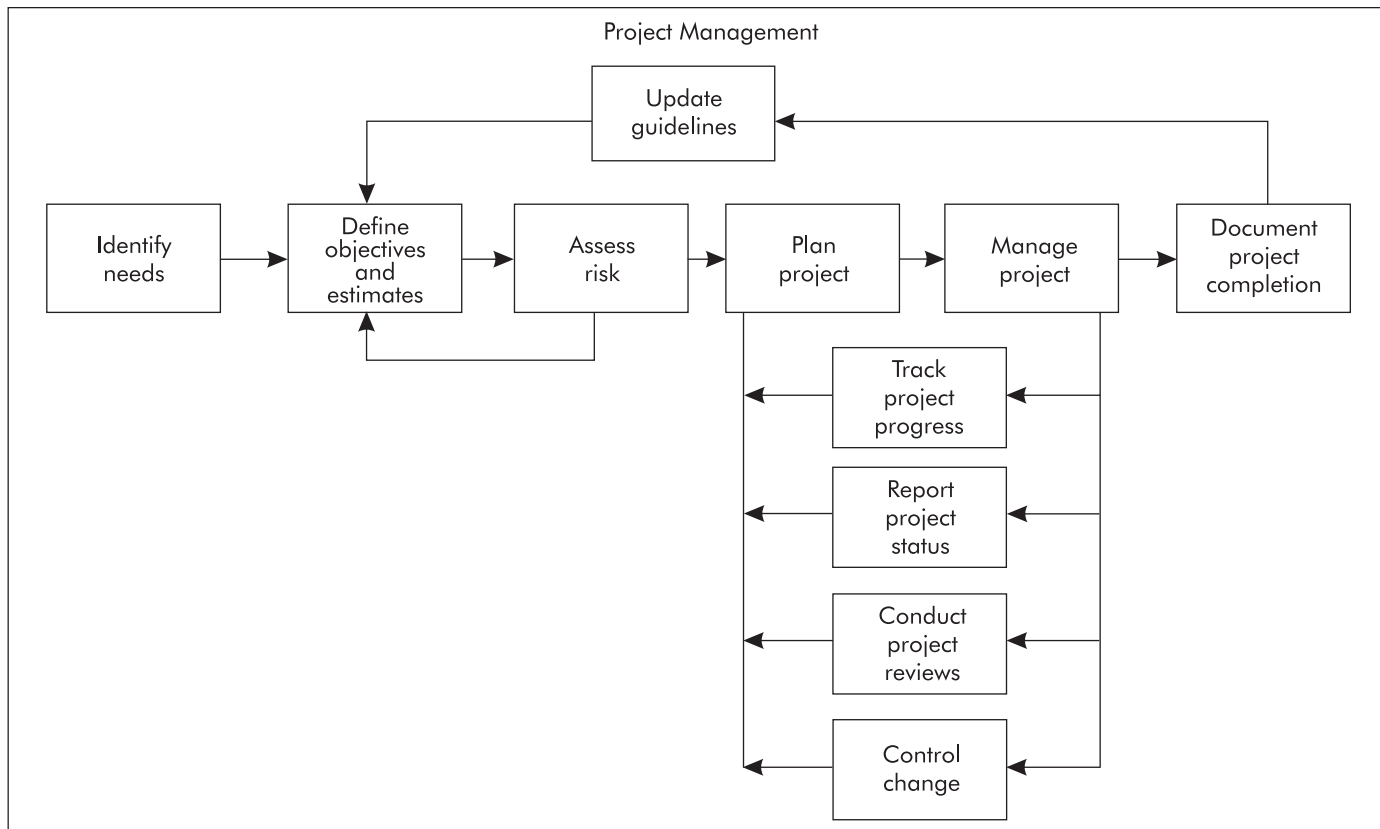


Figure 10-3. Project management process.

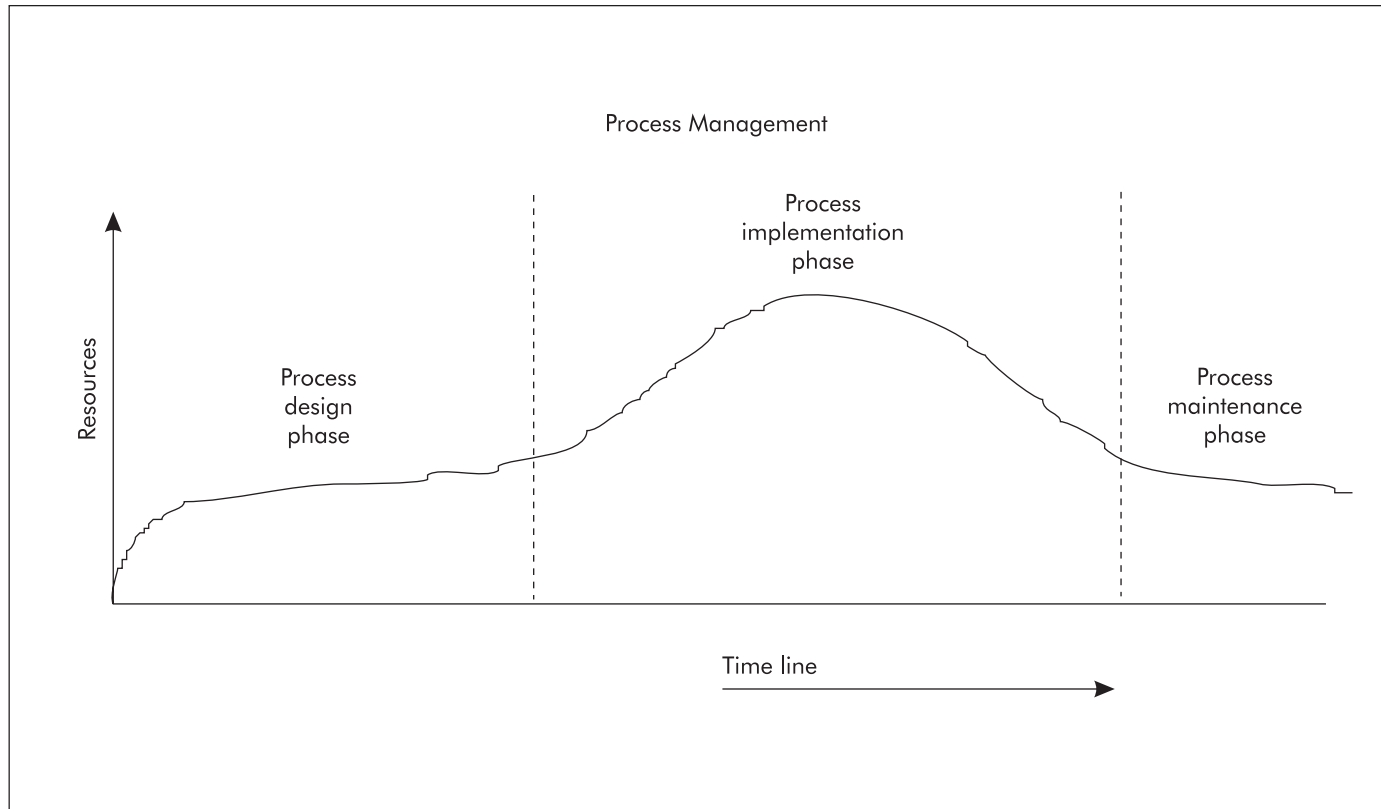


Figure 10-4. Process management chart.

Here we can see that the phases begin with limited resources, ramp up to a set of maximum resources being applied, and then return to minimal resource requirements.

During the *process design* phase, there are myriad small activities taking place, including documenting the process objectives, setting up resource and system requirements, and determining internal and external design requirements.

In the *process implementation* phase the activities include creating a variety of test plans; implementing development procedures; and installing equipment, personnel, and training procedures.

The final phase, *process maintenance*, includes documenting processes and operational systems that are needed to keep the process operating until such a time that the process may no longer be required.

Product Development Time Line

The most common method of charting product development activities today is the time line chart. Normally, concurrent engineering concepts are employed, and this chart design allows instant visual tracking of those concurrent activities. The easiest way of depicting these are through the use of bar graphs similar to those shown in Figure 10-5.

The activities shown in the left-hand column are typical of those required for a normal product development process. Looking at the time-line bars you can see that there is some overlap of activities. This is referred to as *concurrent engineering* and simply indicates that some activities can be started before prior activities are finished. An example of this is the *select material* activity. It can, and must, begin before the *formal design* activity is completed. And, the *build tooling* activity does not have to wait until the *design tooling* activity is totally completed. There is much front-end work that can begin while the tool design is still being performed. Of course there are some activities that depend on results of prior activities before they can begin. An example of this is the *debug tooling* activity. It would indeed be difficult to debug the tool before the tool is completed under the *build tooling* activity.

The time line chart is especially useful because it tells at a glance the number of activities required, gives a description of those activities, the estimated time needed to complete each activity, when the activity will start and finish, and the degree of dependency of that activity on any other activity. Resource figures could be incorporated to show the estimated funding and other resources needed to perform the activities. All information can be updated easily at any time, and checkpoints could be incorporated to ensure these updates.

Why Projects Fail

In an ideal world every project would be successful. But in the real world, some projects fail. By understanding the failure mechanisms we can minimize the

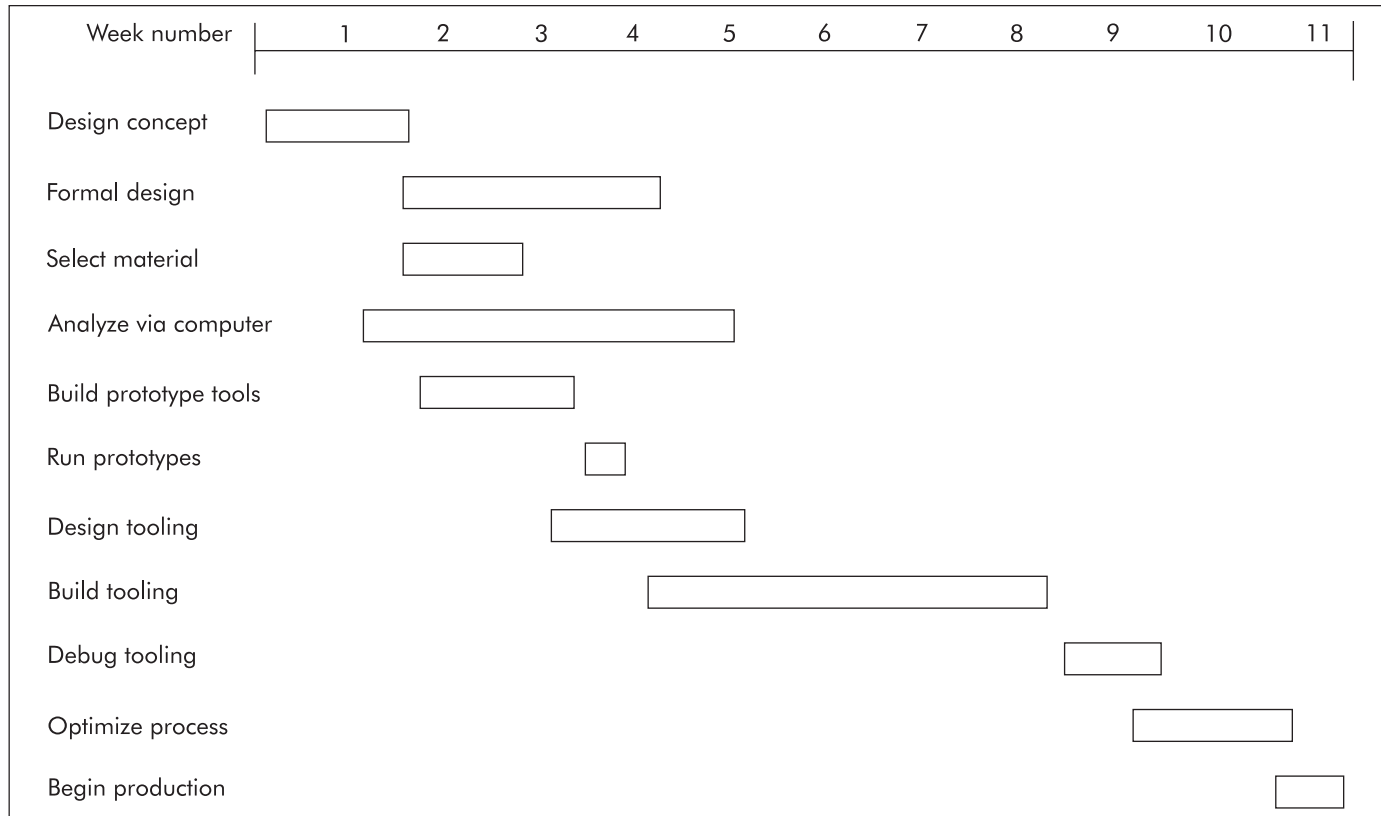


Figure 10-5. Product development time-line chart.

probability of failure for any specific project. Following are some of the more common reasons for failure.

Uninvolved Users

In this case, the end-user requirements have not been properly considered. A cursory study may have been made, but in the interest of time, usually the end user has not had sufficient input as to the product's function, shape, or fit. What develops is a product that the designer assumes the customer (end user) will want. Unfortunately, if the end-user requirements have not been properly solicited, chances are the customer will not be interested in the product because it probably does not meet their company's needs.

Undisciplined Management System

There are times when management takes a passive role in project activities. Sometimes they tend to depend on the individual to perform his or her duties and respond to his or her responsibilities without intervention. While this may seem a noble thing to do, it can be disastrous to the success of a project. Management needs to be involved if only to keep everything moving in the right direction. Those employees who need no supervision still need someone to coordinate everyone's efforts. Without that coordination, any project is doomed to failure. Someone must be designated as the focal point—a person to which everyone else turns for direction. That person need not be a member of management but at least needs to be authorized by management to have control over all aspects of the project. This creates a discipline needed to ensure project success.

Ineffective Communication

If there is no requirement for formal communication between project members, there can be no team control of the project. Each member may be performing his or her own duties quite well, but every project is a team effort and each member of that team must be very aware of what other members are and are *not* doing. All the roles are intertwined and each member depends on the success of other team members. While verbal communications are acceptable as a temporary measure (and in fact should be encouraged), written communications are intended, and expected, to be more formal and more accurate. Time should be taken to select the correct verbiage, appropriate grammar, and proper spelling to ensure that the communication document is not misinterpreted. Because this is such an important facet of project success, all team members should take a course in effective communications prior to being accepted to a project team.

Uncontrolled Changes

During the evolution of a program or project there are bound to be changes of some sort. These are usually focused in the area of product design, material

selection, process selection, and time schedules. These changes must be implemented using a structured, controlled procedure or they may not be identified to the proper personnel responsible for executing them. Also, as additional changes occur, they may not be placed in proper arrangement, giving rise to complete confusion. In the case of product drawing changes, it is not uncommon to find one vendor working with down-level drawings while another is working with up-level, hand-marked drawings. With no structured control there is no guarantee that vendors are working with similar information. It is true that structured control usually does add time to the basic schedule of activities that must take place within a project, but lack of controlled changes will add much more time to that schedule.

Inappropriate Technology

During the Phase 0 status of a project, a decision on which method the product will be manufactured under may end up being based on what is immediately available rather than what is most appropriate. This might happen, for instance, if a company has recently invested a large amount of money in specific equipment that is not being utilized. At first glance it may seem logical to manipulate the product design or alter other requirements to take advantage of that idle equipment. However, such a decision should be scrutinized carefully, as it often results in alterations to the product that make the product less attractive to the end user, thus reducing potential sales. Or the cost of using the equipment, which may not be suited to the application, is so high that the project fails as a result of exorbitant manufacturing costs. It may even be possible that, during the manufacturing process selection activity, there is no one with the proper expertise on the project team. In such cases, only processes that are familiar to the members will be considered, while a more appropriate process may be available, but unknown.

Poorly Defined Products

Finally, poor product design or definition will surely result in project failure. If steps are taken to bypass customer requirements for the sake of reducing the cost of the product, the result could be total failure of acceptance by the user. Or if environmental issues are not considered you may end up with a product intended for out-of-doors use that degrades quickly as the result of improper material selection. This may be due to the simple fact that no one had the foresight to tell the material supplier the product would be used out-of-doors. While it is true that too many notes on a drawing can spoil a product design, it is also true that the lack of proper notes can have the same effect. Care must be taken to ensure that the proper information is communicated regarding the form, fit, and function of a product, especially in the early design stages.

SUMMARY

The product development process starts with an individual idea or concept. This soon develops into a full-fledged project requiring the assistance of other individuals to bring the idea to fruition. At this phase of the project, a brainstorming session is in order. During this session many thoughts, ideas, concepts, concerns, and criticisms are generated, negotiated, and developed.

Although too many people can spoil the success of a project, it is necessary to have proper groups represented by individuals who are responsible for transferring information both to and from the core group of team members needed to develop the product.

The groups that should be represented at all team meetings include Product Design, Purchasing, Material Engineering, Manufacturing Engineering, Quality, and Marketing.

Timetables and levels of responsibility must be established during the first team meeting and adhered to during the entire development process. They may be adjusted as needs require but must be maintained.

One of the early questions to be answered is: Where will the product be made? The answer must take many items into consideration including worker expertise level and in-house capability. If a vendor is chosen, that vendor must meet rigid requirements established by the team.

Volume drives cost. Economies of scale mean that the costs to manufacture and develop a product are directly determined by the number of units estimated to be produced annually. The more products produced, the lower the individual cost.

Any project must be carried through a minimum of eight separate phases starting with an initial proposal, working into production, and finishing up with final termination of the program (usually at the “end of life” of the product being manufactured).

Projects fail for one or more of six basic reasons, ranging from uninvolved users to poorly defined products.

QUESTIONS

1. What groups are typically represented at an initial brainstorming session?
2. Name three questions that must be answered during the brainstorming session.
3. Why must storage be considered when determining manufacturing costs?
4. What do the initials JIT stand for concerning a manufacturing concept?
5. Why should a product development chart be used?
6. What happens during Phase 0 of a product development cycle?

7. During which phase of the product development cycle does the product actually get designed?
8. During which phase of the product development cycle does production actually start?
9. What is the first step in a typical project management program?
10. Name two reasons why a project might fail.

Determining Product Requirements

11

DEFINING THE PRODUCT

Although it may be obvious, it must be stated that a product should be properly defined early in the development process to ensure correct selection of materials, processes, and testing procedures. Unfortunately, in most cases product definition is not accomplished early enough, making constant engineering changes (and subsequent cost overruns and time delays) inevitable. A “rushed” concept or hurried design will guarantee downstream interruptions, missed communications, and lengthy delays in the total development cycle. This does not mean that changes cannot be tolerated. In fact, changes should be expected as the product design evolves from the original proposal to the final production level concept. However, these changes should be planned, negotiated, and properly considered by all development team members and representatives instead of simply being thrown together to meet a schedule or cost limitation.

Defining the product includes activities that place ideal requirements on the product and practical tolerances on those requirements. For instance, a design engineer may wish to have a product that will function properly between the end-use temperatures of -20 to 140°F (-29 to 60°C). This is a common temperature range and many materials will function within these limits. But if the designer does not have confidence in the molder of the product, or if he or she feels that a safety factor must be used to ensure proper product functioning, he or she may exaggerate the requirements and set the temperature range to other limits, e.g., -50 and 250°F (-46 and 121°C). By doing so, the designer may feel comfortable believing that he or she has reduced the potential of failure of the product, but he or she also has severely limited the number of plastic materials that can be used to mold the product. And, as a result, the cost of raw material has skyrocketed, and the vendor base of molders capable of molding the material has dwindled. This amplifies the problem because a “specialist” molder must be used whose per-hour rate for manufacturing is much higher than that of a production molder. In addition, the processing window for molding has been greatly reduced, due to the material of choice, and this increases scrap and defect potential, both of which increase overall manufacturing costs. Quality costs may increase due to having to develop a nonstandard test because

of the nonstandard temperature range requirements. And, finally, the cost of the mold itself may be two to three times normal in order to be built to higher standards and of superior materials to mold the selected plastic material.

This example describes what can happen when a single item is exaggerated, and there are many, many items to be considered when defining a product. All of these added budget and schedule costs may be incurred simply because a designer does not have confidence in a molder. This usually happens because the designer has no contact with the molder during the entire development cycle. There can be little confidence without contact. So it becomes apparent that early involvement by everyone concerned with the success of the program is vital to that success.

In defining a product, the designer usually holds total responsibility. However, the successful designer will make use of every tool possible to create the proper definition. That includes material engineers (not just material selection data bases), tool engineers, process engineers, manufacturing engineers, quality assurance specialists, purchasing agents, design guides (available from material suppliers), and consultants, when necessary. It also includes utilizing the expertise of any vendors selected to work on the program. While the designer is not obligated to act on any of the recommendations provided by this elite group of experienced individuals, he or she would be remiss not to take advantage of this storehouse of knowledge.

TYPICAL PROJECT EXAMPLE

Let's take a look at what might happen during the development cycle of a typical product.* The project consists of developing a penholder that can be placed in the cradle of a desk telephone. The original concept is based on the need for an easy-to-find device for making notes in the middle of the night. But, as we will see, group involvement carries the concept to other areas and creates new business opportunities. In addition, the involvement of vendors helps keep the product manufacturable and inexpensive.

George Byrnes woke up in the middle of the night, switched on the bedside lamp and answered the ringing telephone. His brother was on the other end calling long distance and wanted to give George the flight information for his arrival at the airport in a few hours. George had offered to pick him up once his brother had made the final arrangements for this emergency trip. While his brother began to rattle off the flight numbers and times, George was scrambling to find a pen. He kept a pad of scratch paper at his bedside, but could never seem to

*An actual case study. Though names and certain details have been changed, the facts are accurate.

keep a pen in place. His brother was waiting impatiently on the other end, and the first leg of his flight was being announced. Finally, George tried to memorize the information and told his brother to go ahead so he would not miss his flight. Needless to say, George did not arrive at the right time, or at the right gate, and his anxious brother was kept waiting quite a while.

Phase 0

As a result of this episode, George came up with the idea of providing a special pen and penholder that could be kept at the bedside for taking notes. This was the original concept—just a thought about a product that could be designed to fill a need. Over the next few days George came up with a basic pen-and-chain device that could be fastened to a night stand. The device would keep the pen from being taken away and was similar to the device used to keep pens at the convenience stands in bank lobbies.

Phase 1

George was the owner of a small manufacturer's sales representative firm, and brought his idea into the office one day to tinker with it. He solicited advice from a few of his sales people and things began to roll. What he found was that the other members came up with great, practical ideas on the design of the product. First, the device did not have to be a standard pen. It could be a much shorter version, quite useful for emergencies, but not as appealing for use elsewhere. Making the pen less appealing would reduce the tendency to take it away from its intended area. Second, instead of simply laying it down on the night stand, it could be "snapped" into a small receptacle that was placed on the table top. And, third, it could have adhesive backing and could be stuck to the cradle of the telephone, where it would be out of the way until needed, and then readily handy. On paper the concept appeared as shown in Figure 11-1.

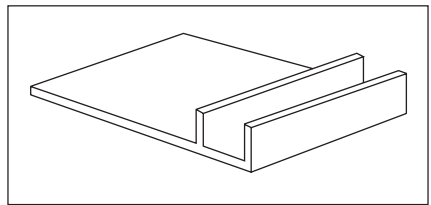


Figure 11-1. Initial penholder concept.

Phase 2

As these ideas flowed back and forth, another thought developed. This device could be used as an advertising tool. Advertisements could be stamped on the flat surface, along with poison control numbers, emergency numbers, and other information thought to be important during times of crisis. These would always be at hand and the included pen would be available in the event that information needed to be written down during conversations. One of the team members even suggested offering a "glow-in-the-dark" version for nighttime viewing because

the adhesive backing would allow the user to place the penholder anywhere beside the intended cradle area of the phone. Soon the product was designed to the satisfaction of the members of George's team. But George, being involved with manufacturing processes, knew that he should solicit advice from manufacturing personnel also. He talked to metalworking shops and plastic working shops. He decided to go with plastic for a variety of reasons, but mainly because he could get the product molded all in one piece with a flexible "snap" for holding the pen.

Phases 3 Through 8

George contacted a consultant, who put him in touch with a molder in the area. The molder discussed the device with George and showed George how some minor changes in the basic design would help make the device more manufacturable. He also discussed the material choices with George. The molder brought along his tool designer to assist in determining how many cavities should be made, and how the cost of the mold could be kept at a minimum without adversely affecting the quality of the product. It was suggested that a single-cavity prototype mold be built to prove

the mold design and provide some parts for marketing and testing purposes. Then the production mold could be built utilizing the information provided by the prototypes. This was agreed to and the manufacturing process began based on the design shown in Figure 11-2.

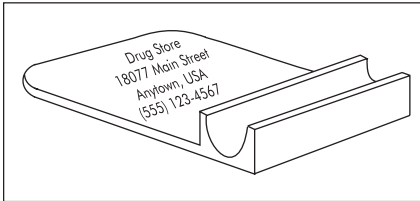


Figure 11-2. Final penholder design.

The project was a huge success.

Several million of the penholders were sold as advertising tools for a variety of firms including doctors' offices, funeral homes, hardware stores, bookstores, churches, and restaurants. Approximately 15% of the devices were made to glow in the dark through the use of a phosphorescent pigment added to the plastic material. The basic material was a commodity resin (polypropylene) which kept the manufacturing costs low. The molder drop-shipped the products directly to the buyer, which eliminated middle-man freight costs. The pens were purchased from a major pen manufacturer at a very low cost and shipped to the molder who utilized a press operator to assemble and package the units during the molding cycle. Every aspect of the product design, material selection, tool design and build, and manufacturing was carefully thought out, detailed, negotiated, and resolved through the efforts of early involvement by critical team members. And the entire cycle, from the night of George's brother's phone call to the day the first parts were produced on an eight-cavity production mold, took a total of 16

weeks. This shows that proper, early involvement need not mean delayed development. This project took place in the early 1980s. With the computerized technology available today, that total time could be greatly reduced.

DETERMINING ACTUAL REQUIREMENTS

There are seven basic areas of interest to consider when determining the actual requirements of a product to be manufactured. These are the functional, physical and mechanical, thermal, and electrical aspects; flammability rating; weather and exposure limits; and safety and environmental issues. We examine each in the following paragraphs.

Functional Aspects

In the *functional* area we are concerned with what the product is intended to do, and how that is accomplished. In this area we find such items as color, fit, aesthetic appeal, etc.

Color

Color is an important feature of any product that is visible to the end user. Most plastics are available in many standard colors, and most can be custom colored, either internal to the pellets by the original compounder, after original pelletizing by a compounding specialist, or by the molder just prior to molding the pellets. If the material is to be colored after original pelletizing, it is done so through the use of either a pigment or a dye, pigments being the most common because they are available in powder form and easy to use. Dyes are also used but they tend to be messier and highly concentrated, creating a potential for inconsistent color from batch to batch when mixed with the plastic material.

It is very common, especially on short runs (less than 50,000 pieces) for the molder to add the color at the molding process. This is normally done by purchasing a pellet form of the base material to be colored, but in highly concentrated versions of the color desired. These pellets are called “concentrates,” and the molder can vary the shade or hue of a particular color simply by using more or less of the concentrate mixed with the base plastic. The concentrate is added at a rate of approximately 1 lb (0.45 kg) to 100 lb (45 kg) of base resin. When this mix travels through the heating cylinder of the molding machine, the concentrates melt right along with the base pellets and both are homogeneously combined through the mixing action of the injection unit’s screw. This creates a well-blended mixture of the final, intended color. If the molder is using a plunger machine instead of a screw machine, the pellets still melt but are not blended together and will create a marbled, or swirled, effect in the molded part. If that is the intended appearance, a plunger machine is the only way to achieve it as a molded-in effect.

Because virtually millions of color combinations are available, selecting the correct one for a specific product can be a formidable task. In most cases a color chip sample, available from the material supplier, can be utilized, although there is a degree of risk involved. The color chip sample may be molded in a material other than the exact one selected for the product, so the color may appear different on the sample versus the color of a product molded in the selected material. Also, the color chip may not have the exact texture of the product being molded. This may result in a totally different appearance of the molded part, versus the color sample, due to different optical properties of the two. To ensure accuracy, ask for a color chip selection molded in the exact material and surface texture intended for the final product.

Finally, the exact shade, or hue, of color desired may change as a result of processing parameter differences, especially concerning heat. A hot plastic will provide a slightly different shade of color than a colder plastic. And a hot mold will do the same over a colder mold. These differences are especially noted immediately after the part is ejected from the mold compared to a part that has cooled for an hour or so. For this reason, it is imperative that mating parts (from different molds) be molded at the same material and mold temperatures during processing, and that the color not be checked for at least an hour after molding. Only then will consistent, accurate color matching be possible.

Fit

With fit, we are concerned with dimensional characteristics of the molded part: tolerances, shrink factors, hole locations, and similar issues related to dimensions.

Tolerances. To begin with, we must consider tolerances. Obviously, it is not possible to design a product without the use of dimensions. Not quite as obvious is the fact that it is not possible to manufacture a product to the *exact* dimensions placed on a product drawing. This is because of changes in environmental conditions, wear of equipment and tools, and the variations in the ability and expertise of the people building the molds and operating the molding equipment, even if computers are used for control. Therefore, the product designer must be willing to incorporate tolerances for these drawing dimensions. Because there are many differences between all of the more than 20,000 available plastic materials to choose from, each one performs slightly differently concerning the ability to control dimensions. Thus, the product designer must be aware that some materials are capable of being molded into parts to very tight tolerances while others are not. The key to proper tolerances is for the designer to request only the amount of tolerance needed for the specific application. The reason for this is that tighter tolerances cost more money to produce.

Table XI-1 lists several plastic materials and shows how some are more capable of controlling dimensions than others. This table lists various common

Table XI-1. Recommended Tolerances for 6-in. (152-mm) Dimension

Material	Preferred (\pm)	Tight (\pm)
ABS	0.011 in. (0.279 mm)	0.0055 in. (0.140 mm)
Acetal	0.020 in. (0.508 mm)	0.010 in. (0.254 mm)
Acetate	0.0145 in. (0.368 mm)	0.0055 in. (0.140 mm)
Acrylic	0.0145 in. (0.368 mm)	0.0055 in. (0.140 mm)
Nylon (6/6)	0.020 in. (0.508 mm)	0.010 in. (0.254 mm)
Polyethylene	0.030 in. (0.762 mm)	0.011 in. (0.279 mm)
Polypropylene	0.026 in. (0.660 mm)	0.010 in. (0.254 mm)
Polycarbonate	0.011 in. (0.279 mm)	0.0055 in. (0.140 mm)
PVC (rigid)	0.011 in. (0.279 mm)	0.0055 in. (0.140 mm)
Styrene	0.0145 in. (0.368 mm)	0.0055 in. (0.140 mm)
Polysulfone	0.011 in. (0.279 mm)	0.0055 in. (0.140 mm)
PPO-modified	0.011 in. (0.279 mm)	0.0055 in. (0.140 mm)
Polyester (TP)	0.020 in. (0.508 mm)	0.010 in. (0.254 mm)
Glass-reinforced Materials		
Shrinkage of 0.001/0.004 in./in. (0.025/0.102 mm/mm)	0.0145 in. (0.368 mm)	0.0055 in. (0.140 mm)
Shrinkage of 0.005/0.008 in./in. (0.127/0.203 mm/mm)	0.020 in. (0.508 mm)	0.010 in. (0.254 mm)

materials down the left column and shows typical tolerance requirements for a 6-in. (152-mm) dimension in the two right-hand columns.

The “Preferred” tolerance is that suggested by the Society of the Plastics Industry and reflects the amount of tolerance needed for standard molding costs. If the “Tight” tolerance column is used by the designer, the molded parts will cost substantially more to manufacture. The correct, exact tolerance for any specific plastic can be found in the material supplier’s information bank.

Shrink factors. In addition to tolerances, the designer must be aware that there are differences in how parts will shrink, based on the material being molded. In Chapter 1, we discussed shrinkage factors and how they affect the molded product. Remember that *all* materials (except water) will expand when they are heated and contract (shrink) when they are cooled. Because we are heating the plastic for molding, the mold into which we inject the plastic must be machined slightly larger than what we want for final dimensions on our molded product. This is done to allow for the expansion of the plastic while being molded and the shrinkage of it when it fully cools. Not all of the cooling takes place in the mold. To fully cool the plastic in the mold, the mold would have to stay closed for days, which is not practical or cost effective. Therefore, we eject the part after a normal cycle of usually less than a minute. The part has formed a hard,

cooled, skin on the surface which will hold shape until the entire cooling process is finished. But during the cooling process, the part will continue to shrink slightly. If all the walls and details of the product have exactly the same dimensions and thicknesses, the part will usually shrink uniformly. But, as with most designs, the product contains varying wall thicknesses, bosses, rib dimensions, etc., and the part may shrink in a nonuniform way, resulting in warpage, bowing, and dimensional variations from part to part.

There are three basic groups of shrink factors for thermoplastic materials: *low*, *medium*, and *high*. Low shrink factors are considered those between 0.000 in./in. (mm/mm) and 0.005 in./in. (0.127 mm/mm), medium are considered between 0.005 and 0.010 in./in. (0.127 and 0.254 mm/mm), and high are considered those between 0.010 and 0.015 in./in. (0.254 and 0.381 mm/mm). Some materials are found to have shrink factors greater than those listed, but they are not pertinent to this discussion. The exact shrink factor for any specific plastic material is available directly from the manufacturer of that plastic. The greater the shrink factor, the more difficult it is to control the dimensions of parts molded in that material. Thus, greater tolerances would need to be employed.

Appearance

Appearance is a critical area of consideration for any molded product that will have surfaces or features visible to the user. Such items as ejector pin witness marks (or protrusions or depressions), gate stubble, flash, and discolored areas may be offensive, or (as in the case of sharp flash) even dangerous to the user. Good design practices will help minimize these concerns.

General. The designer can note on the drawing that markings, such as those left by ejector pins and gate stubble, should not be allowed in critical areas. Bear in mind that the part must be gated somewhere and must be ejected from the mold, and *witness marks* of some type will be evident. But these can be placed in areas other than those deemed critical by the designer. Also, if flash would be a cause of failure or defect if located in specific areas, those areas should be identified on the drawing. The mold designer will then be alerted and aware of the potential problem and design the mold such that no flash would be created in those areas.

Appearance of a molded part is directly dependent upon the type of material selected, and the processing parameters used for molding the part. In addition, the surface of the part will duplicate the surface of the mold producing the part. If a specific finish is required (such as a certain texture, or optical finish) it must be specified in detail on the drawing. The material supplier should be contacted to make sure the selected plastic is capable of reproducing that finish in the molded part.

Texture. Textures are sometimes desired for aesthetic purposes and range from a fine satin finish to a coarse alligator hide. They can be used for hiding

mold and process imperfections such as ejector pin marks, knit lines, sink marks, or blush. On the molded product they help hide fingerprints, dirt, and scratch marks.

If texture is to be considered, there are some definite issues that must be addressed. First of all, to select a texture requires access to samples of thousands of standard *available* textures. There are texture engravers (and etchers) who specialize in placing texture finishes on mold surfaces. They can be found in any good buyers guide oriented toward the plastic industry. The engravers will provide samples of textures that are in the general area of what the designer defines (such as coarse, medium, or fine). Then the designer selects a texture from the samples and places the specific texture code number and “provider” data on the drawing as part of the product design. The area to be textured is defined on the drawing also. It is a good idea to leave a “flat” area (0.030 to 0.060 in. [0.762 to 1.524 mm]) around the textured area to minimize “hanging-edge” conditions of the texture engraving. If hanging-edge areas are present they may cause excessive flashing or early wear on the mold surface.

The selection of a texture from a sample variety is subjective to say the least. The designer is the one to make that selection. But a warning must be given: the selected sample may not reflect the true texture appearance of the final molded part. This is because the sample is usually provided on a flat black plastic that might be vacuum formed. That same texture on a glossy red plastic that is injection molded will have an entirely different appearance. Ideally, the designer should examine a desired texture as found on the same material produced under the same process that the finished part will see.

During the molding process, the incoming plastic material will abrade the metal surfaces of the injection mold and eventually wear that surface down. Although the amount of wear is usually minimal (about 0.001 in. [0.0254 mm] after 100,000 cycles for nonreinforced plastics), the effect is more dramatic on a textured surface. This is because the texture is actually a series of engraved (or etched) hills and valleys over which the plastic travels. The abrasion eventually flattens out the hills and tends to fill in the valleys. As this occurs, the textured surface gradually levels and disappears, especially when using glass-reinforced plastics. Texture can be reapplied in worn areas, but it is tricky to match it with the surrounding texture.

Another consideration regarding texture is draft angle differences. If the texture is on a molded side wall of a product, that wall must have an additional amount of draft angle, in the amount of 1 degree for every 0.001-in. (0.0254-mm) depth of texture. Most textures are in the range of 0.0005 to 0.003 in. (0.0127 to 0.0762 mm) deep. Deeper textures are available, but uncommon.

Physical and Mechanical Aspects

In this area we are concerned with items such as method of assembly, strengthening ribs, proper wall thickness, the influence of stress, and structural properties.

Because most parts require subsequent assembly of some type, we start with that discussion.

Mechanical Assembly Methods

Usually, a product can be considered complete only after it is joined in some way to other parts. The plastic part being molded may be an internal component of a larger product. Or the molded part may have other components assembled within, or to it. If assembly is to be performed, proper assembly methods must be followed.

Mechanical assembly can be performed through a variety of methods. One of the oldest methods of assembling plastic products is by using screws, or, in some cases, bolts. With this method, specific mounting considerations must be addressed. The following illustrations show some of these considerations. First, we look at an improper bolt assembly design.

The result of using the concept shown in Figure 11-3 is that the walls surrounding the assembly bolt will warp due to the bending stresses exerted when tightening the bolt. No mechanical limits are available to keep the stress from being created. Typically, the bolt and nut (and washer, if used) are made of high-strength metal materials and the plastic material becomes overstressed well

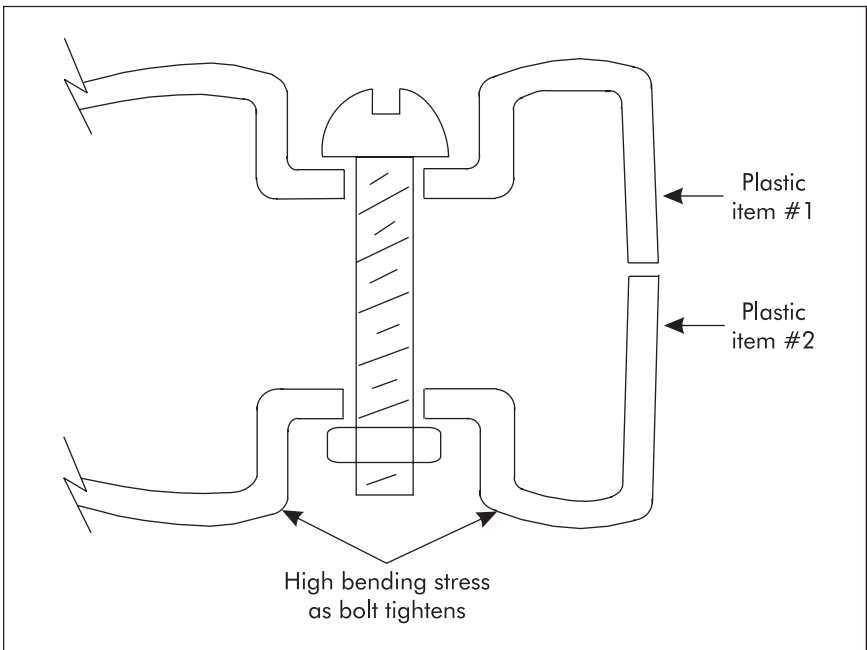


Figure 11-3. Improper bolt assembly design.

before the metal fasteners break, deform, or strip. It is in the best interest of the designer to devise methods that will limit the amount of stress applied to the plastic when the fastening device is tightened. This basic concept can be understood by analyzing the drawing in Figure 11-4.

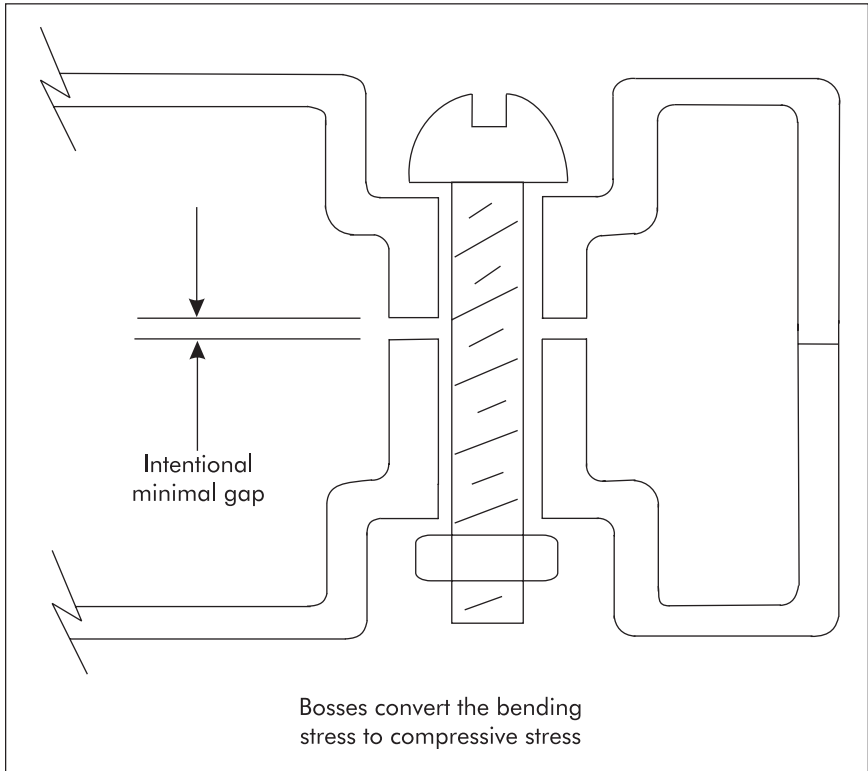


Figure 11-4. Preferred bolt assembly design.

The first thing to notice in this drawing is that bosses are added to both of the parts being assembled. This is done to bring the plastic parts close to each other even before using the fastening bolt and to create a maximum strength condition around the fastening bolt by utilizing a cylindrical shape.

Note that the walls making up the boss and surrounding area are as close to equal as possible. This minimizes the development of stress. Additional strength and support can be achieved by placing ribs between the boss and surrounding walls. The use of these ribs gives increased strength to the boss without using excessive material which would result in extra material cost (which, in turn,

would increase processing costs due to longer cycle time requirements) and cause unsightly sink marks that result from excessive shrinkage in the thicker wall sections.

Also, there is an intentional gap (0.015 to 0.020 in. [0.381 to 0.508 mm]) designed between the faces of the meeting bosses. This gap acts as a “stop” which will convert tightening stress to compressive stress and dissipate that stress through the cylindrical walls of the bosses. This keeps the plastic housing itself from deforming.

Other boss design concepts are shown in Figure 11-5.

These boss designs utilize consistent wall thicknesses and provide maximum boss strength without sacrificing appearance (due to sinks) or physical properties (due to stress). *A* is a design for what is called an “outside boss.” *B* is for a boss at a wall location, and *C* is for an isolated boss (away from a wall location). Whenever possible, supporting ribs should be used on the sides of bosses, as shown in items *B* and *C*.

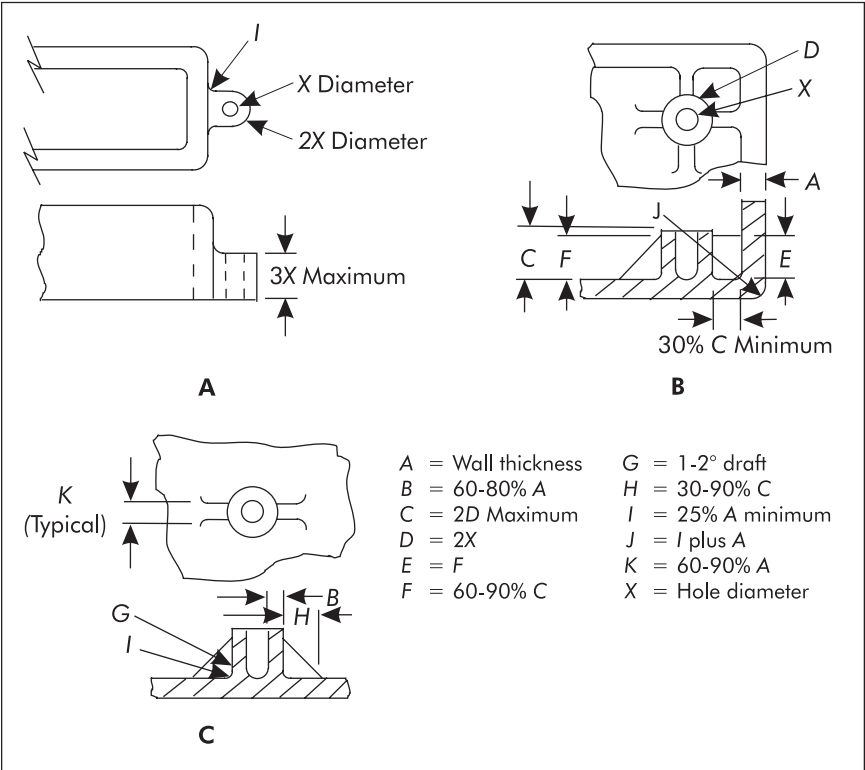


Figure 11-5. Preferred boss design concepts.

If a blind hole is intended in the boss, the core pin forming that hole must be at least long enough to come within the nominal wall thickness of the bottom of the boss. This is shown in both *B* and *C* in Figure 11-5.

Use of screws. If a flat plastic part is being assembled to a metal part (or to another plastic part that has a metal insert) using screws, it is again necessary to consider the torque conditions present in the assembly. Figure 11-6a shows what should be done if the designer wishes to keep the screw head from protruding above the plastic surface.

The flat-head screw design should not be considered because it may cause high stress resulting from the wedging action of the screw head. Instead, the

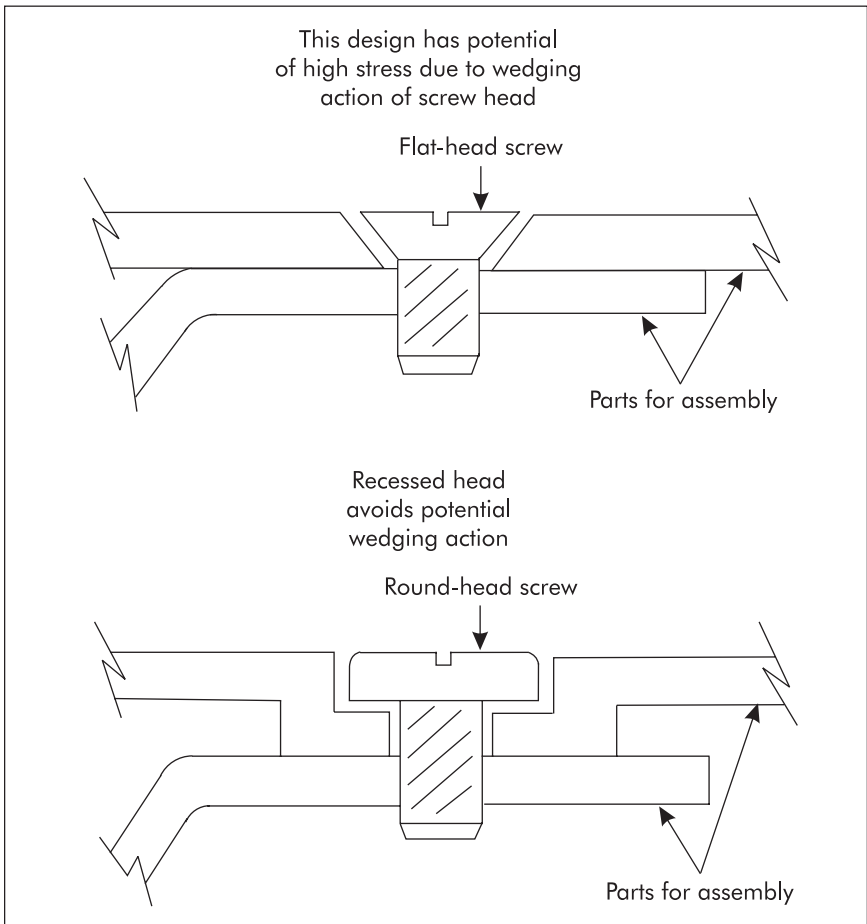


Figure 11-6a. Assembling flat components with screws.

round head (truss) design should be used and the area surrounding the head should be stepped, or recessed, to accept it. This eliminates the potential for wedge-induced stress.

Controlling torque. Excessive torque on an assembly screw (or bolt) can destroy the plastic surrounding the screw and result in failure of the product. This is especially true in a situation where the screw can be tightened in the field, by the user. Special screw designs, such as those shown in Figure 11-6b, can be used to control torque and minimize or eliminate the potential of over-tightening that otherwise could destroy the product.

The use of shoulder screws, flange-head screws, and washer designs will limit the amount of compressive stress that can be applied to a screw assembly area.

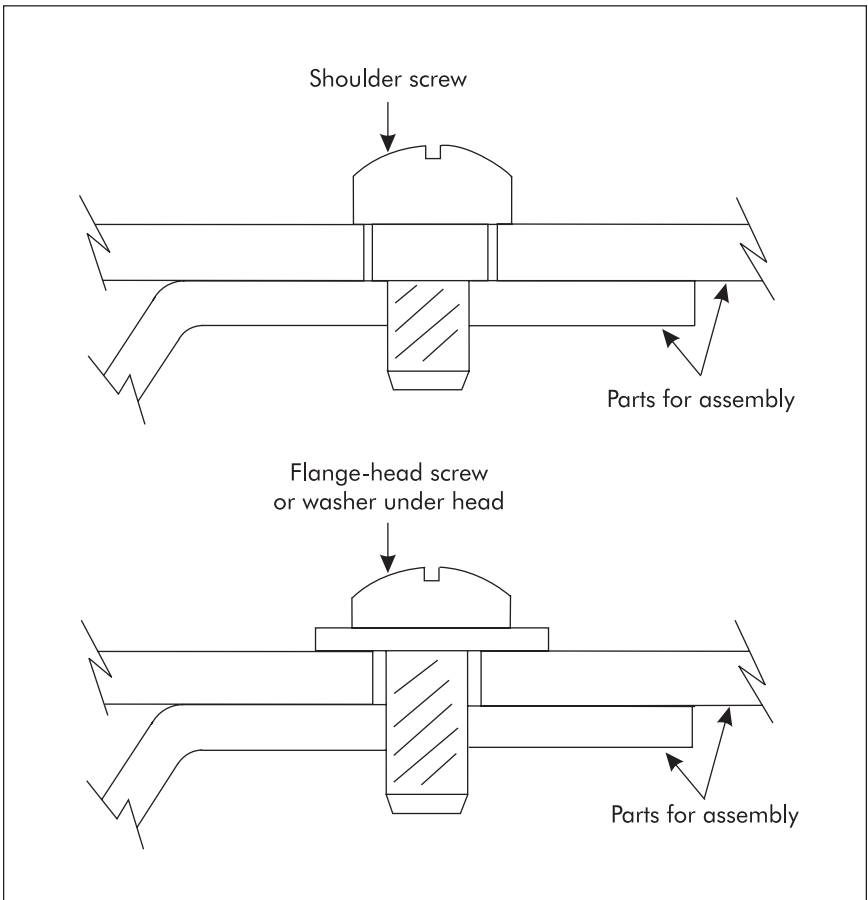


Figure 11-6b. Controlling screw torque.

Retention. Proper thread diameter and size must be used to ensure screw (or bolt) retention. There is also a relationship between the material selected and retention strength. The diameter of the cored hole in the plastic part is critical for proper retention. An example is shown in Figure 11-7a for polycarbonate materials. All of this information is available and should be obtained from the manufacturer of the specific plastic material being used.

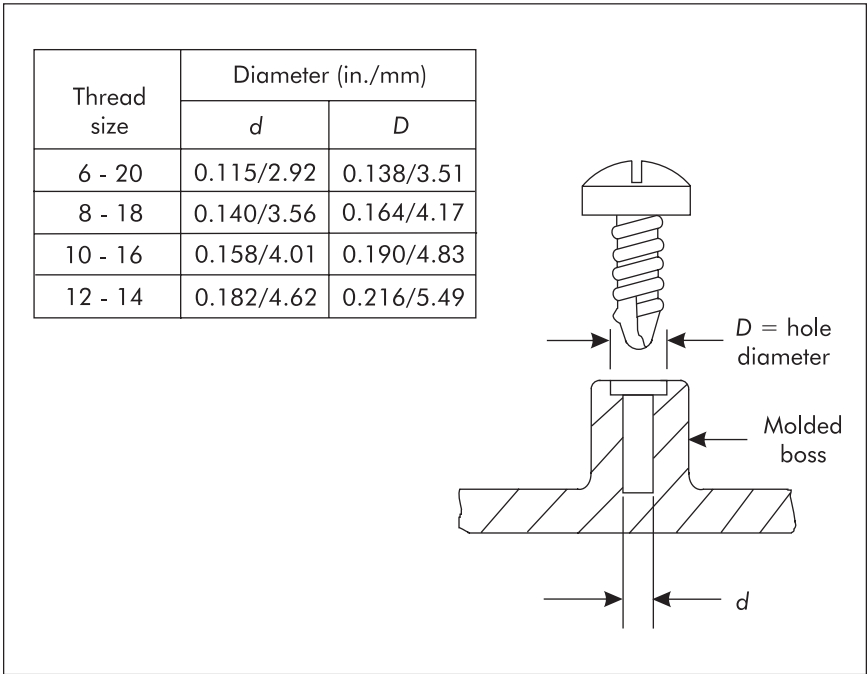


Figure 11-7a. Proper part-to-boss location.

Self-tapping screws. Self-tapping screws are used frequently in the injection molding industry to assemble plastic parts, especially if those parts are intended to be disassembled and assembled repeatedly. To reduce stresses, any part assembled to another by means of a self-tapping screw must be located flush with the top of the self-tapping screw boss, as demonstrated in Figure 11-7b.

Metal inserts. If direct screw-to-plastic contact does not provide enough retention strength, it is good practice to use a metal insert. This can be molded into the plastic (for maximum strength, but increased processing time and cost), or inserted into a cored hole in the plastic after molding. Figures 11-8a and b show typical examples of both conditions.

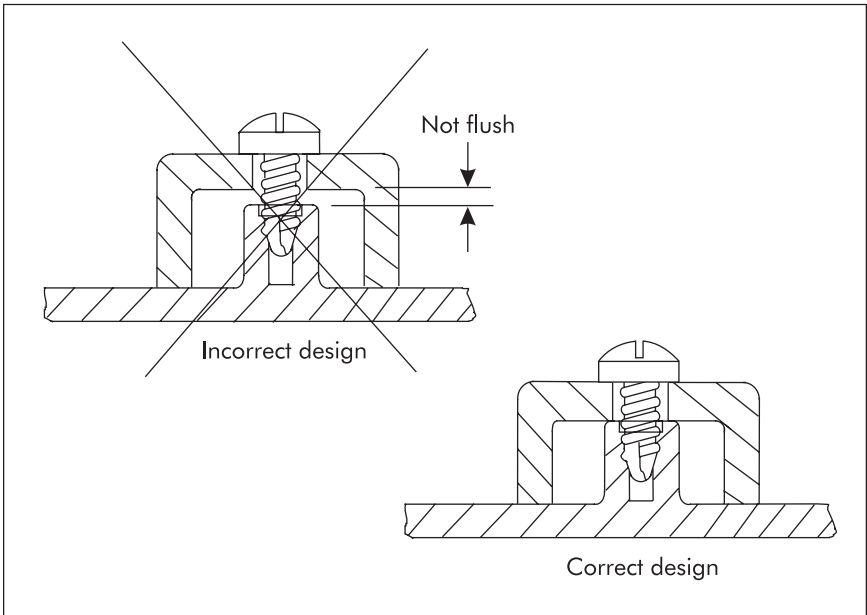


Figure 11-7b. Self-tapping screw and boss design.

If the after-molding insertion method is used, it can be performed using a cold, hot, or sonic insertion process. In all cases the insert is forced into a molded hole. The hole is sized to create a tight fit between the insert and the plastic molding.

When using the cold insertion method, inserts such as those shown in Figure 11-8b are used. These have a feature that allows them to compress slightly while being inserted and then expand after insertion to fit the cored hole tightly.

While this is an inexpensive method, requiring little in the way of equipment, the inserts have only fair holding power in the surrounding plastic. Also, a high degree of stress is created in the area of insertion and this may result in premature failure of the assembly.

In the hot insertion process, the insert is first heated, then pushed into the hole. The hot insert softens the surrounding plastic, which then molds itself around the knurled or otherwise undercut outside diameter of the insert. Upon cooling, the plastic mechanically locks to the insert. The insert designs are similar to those shown in Figure 11-8b, but the cored hole in the plastic part is of a slightly smaller diameter, and the inserts do not need the feature splits that allow contraction for insertion. This results in greater holding power than that found in the cold insertion process. There is, of course, an additional expense with the hot insertion process owing to the equipment needed. Because of the

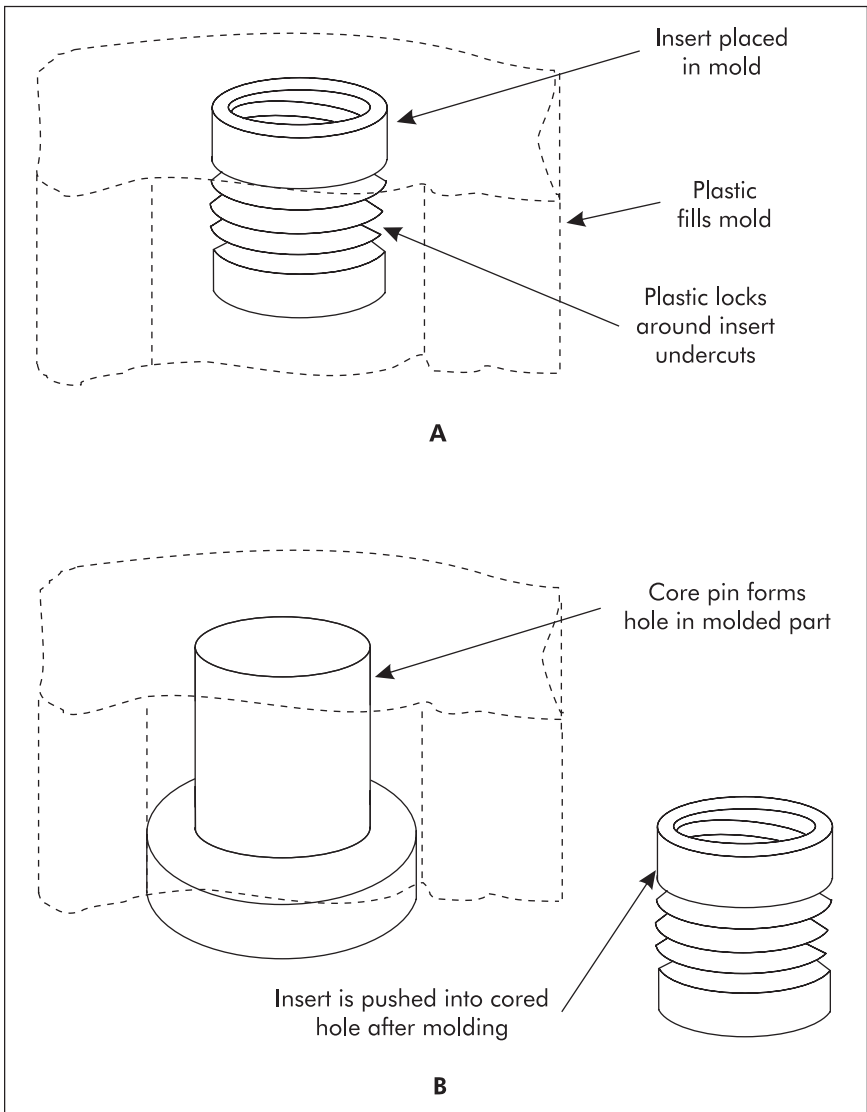


Figure 11-8a. Examples of threaded inserts.

plastic being slightly heated during insertion, the amount of stress created in this method is lower than that caused by the cold insertion process.

If the sonic insertion process is used, a specially designed insert (shown in Figure 11-8c) is pushed into the hole by applying slight pressure to a standard sonic welding horn that focuses sonic energy into the insert.

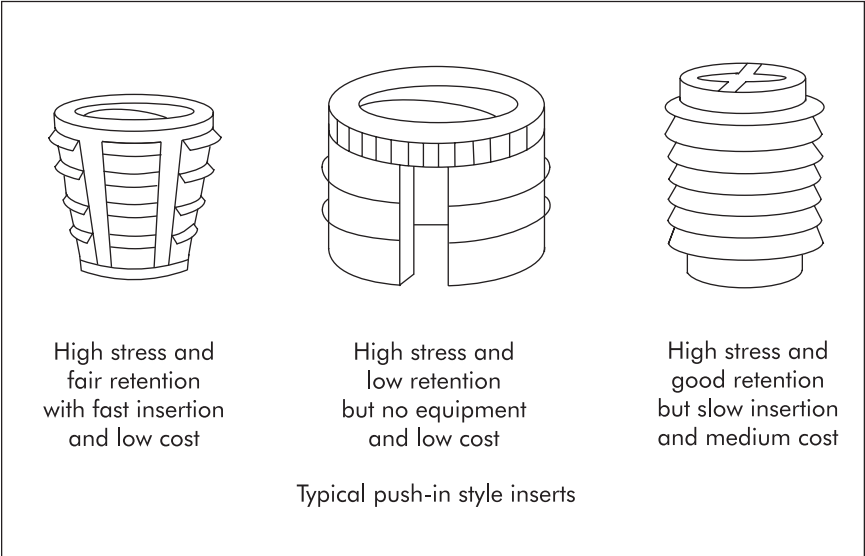


Figure 11-8b. Typical insert designs for cold insertion.

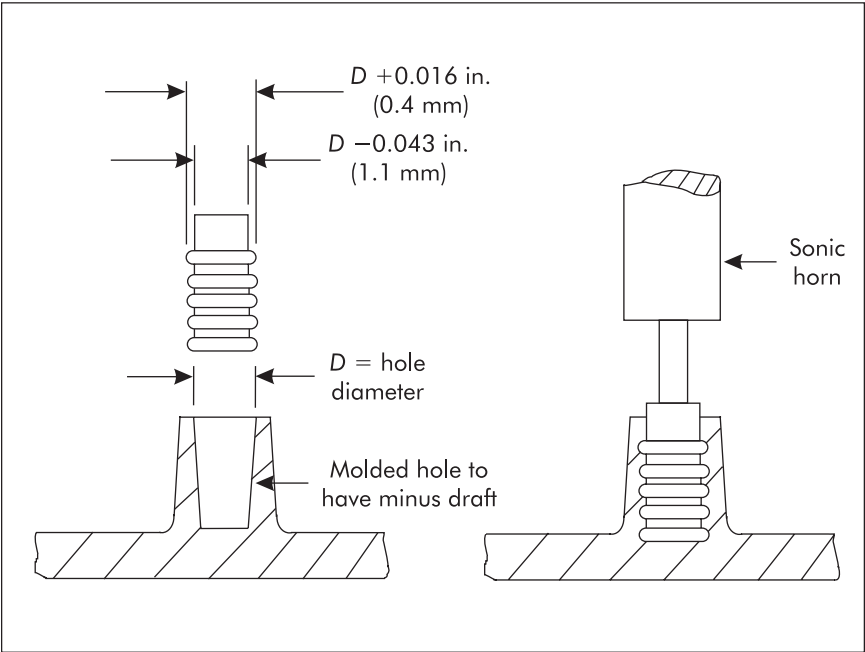


Figure 11-8c. Sonic-style insert installation.

This energy causes a vibration of the metal insert. The vibrating insert creates friction against the surrounding plastic material, making the plastic soften, thus allowing the insert to be pushed into the cored hole. The soft plastic forms around the undercuts in the insert, and upon cooling the plastic is mechanically locked to the plastic. Again, additional expense is involved because of the equipment used, but ultrasonic insertion is the strongest of all after-molding insertion methods. It provides the greatest amount of retention, and because the plastic material is heated to melting, no stress is created by this assembly method.

Rivets and eyelets. Rivets and eyelets are used for holding parts either securely together or loosely, when swivel motion between the parts is specified. These fasteners are generally used only with higher impact materials such as ABS or polycarbonate.

Most rivets are of the “blind” design consisting of a hollow body (shank) and a solid pin (see Figure 11-9). They are set by driving or pulling the solid pin through the hollow shank. This action causes a flaring of the shank on the blind side of the hole, giving the desired locking effect. Eyelets are essentially rivets, but are one piece and do not utilize the pin-in-shank design. Normally, eyelets are used in thinner materials, but serve the same basic purposes as rivets.

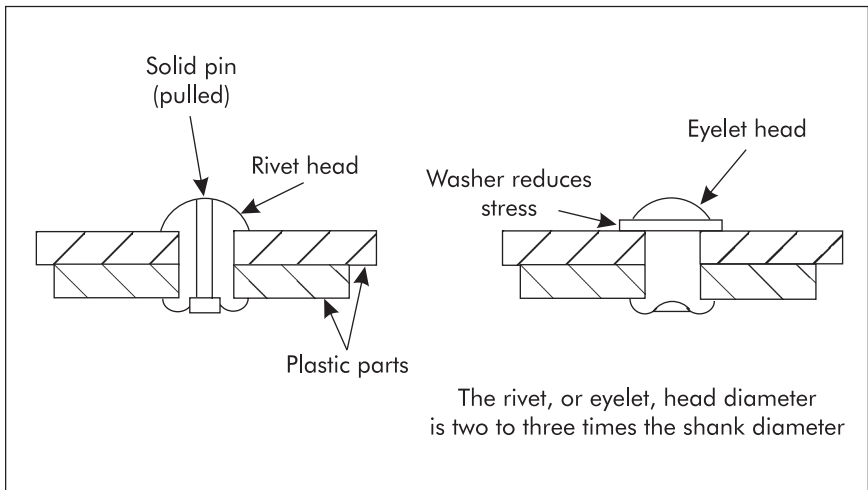


Figure 11-9. Rivet/eyelet specifications.

Living (integral) hinge designs. A “living hinge” is a thin connecting flap located between two plastic parts. It connects the two parts to make them one, integral part. Typically, this device is used on containers consisting of a lid and body. Figure 11-10 shows a typical living-hinge design.

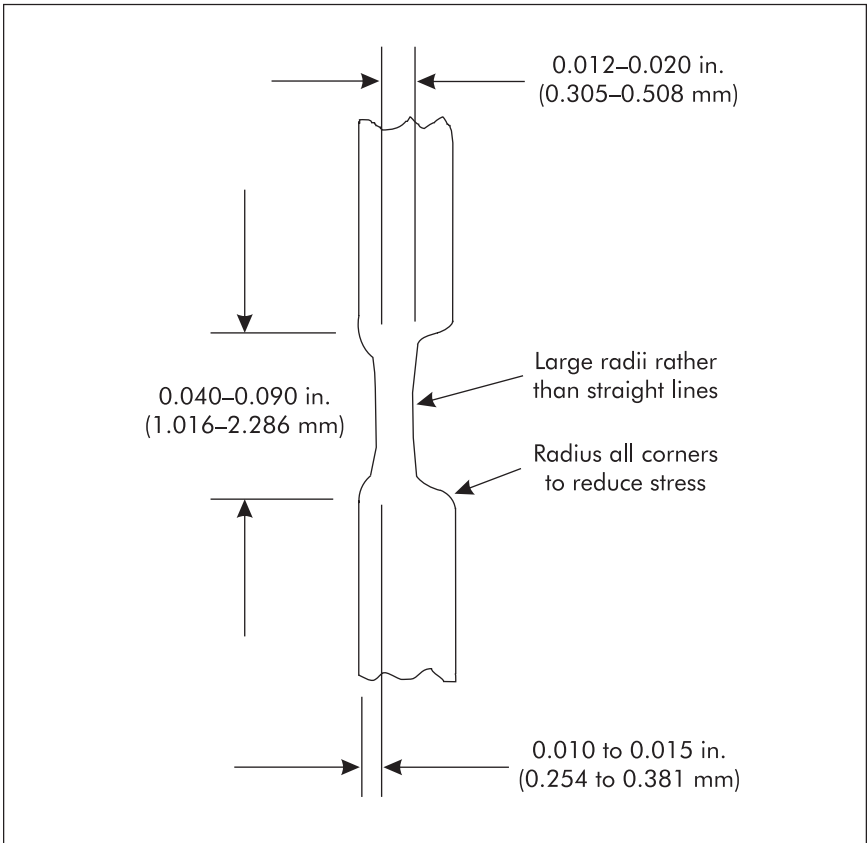


Figure 11-10. Typical living-hinge design.

Successful living hinges can only be considered in products using specific plastic materials that are crystalline in nature and somewhat flexible, such as polyethylene and polypropylene. Success of the hinge design itself is dependent upon proper processing techniques by the injection molder. These techniques include slowly cooling the part in the mold at the correct temperature for the correct amount of time to allow proper crystallinity, and flexing the hinge immediately upon removal from the mold to orient the molecular chains in the proper direction. If either of these processing components is missing, the life of the hinge will be reduced to only a few flexures, then it will crack and tear. A correctly-designed hinge that has been properly molded will last through as many as 300,000 flexures.

The most important area of the living hinge is the flap itself. As Figure 11-10 reflects, the thickness of the flap must be accurately produced to provide a

thickness ranging from 0.008 to 0.020 in. (0.203 to 0.508 mm) with an average of approximately 0.015 in. (0.381 mm). The thicker dimensions allow easier flow of plastic material during injection processing, and can be used when the flexing of the hinge will not approach 180 degrees of arc. When the hinge is used for container applications, the thinner dimension should be used. This impairs flow during the molding process, but allows ease of flexing for the end user.

Pin-hinge design. Pin hinges are used as a more conventional method of hinging plastic components. Figure 11-11 shows the elements of common pin-hinge designs.

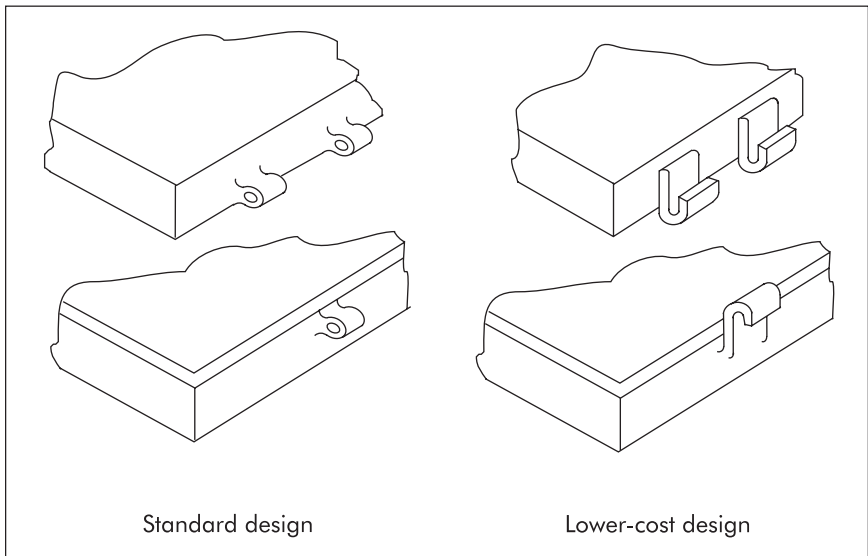


Figure 11-11. Pin-hinge design.

Once the two plastic parts are brought together, a hinge pin is inserted through the hinge devices. The pin then holds the parts in place.

The design on the left is more expensive because it requires cam action coring devices in the mold to form the holes and hinges. In addition, because of the presence of core pins, this design lends itself to the formation of knit lines. Knit lines are normally the weakest area of a molded part so the hinge may be prone to failure from cracking. The device on the right is less expensive because it can be molded in what is known as “draw” position, without the need for core pins. The tooling is less costly, and there is no tendency to form knit lines.

Snap-fit design. Molded-in snap-fit devices replace the use of items such as screws, bolts, rivets, or pins to assemble plastic parts. Snap-fit devices only

require that the plastic snap be able to flex slightly and return to its original position. All rigid and semirigid thermoplastics can utilize snap-fits, including the more rigid polycarbonates. The most common type of snap-fit design (cantilever beam) is shown in Figure 11-12.

Note that the snap-fit arm (lever) is slightly tapered. This design distributes the deflection stress over the entire length of the latching arm (or beam). A straight arm results in the deflection stress being concentrated at the base of the latching arm, which may lead to failure through cracking. A latch with a tapered

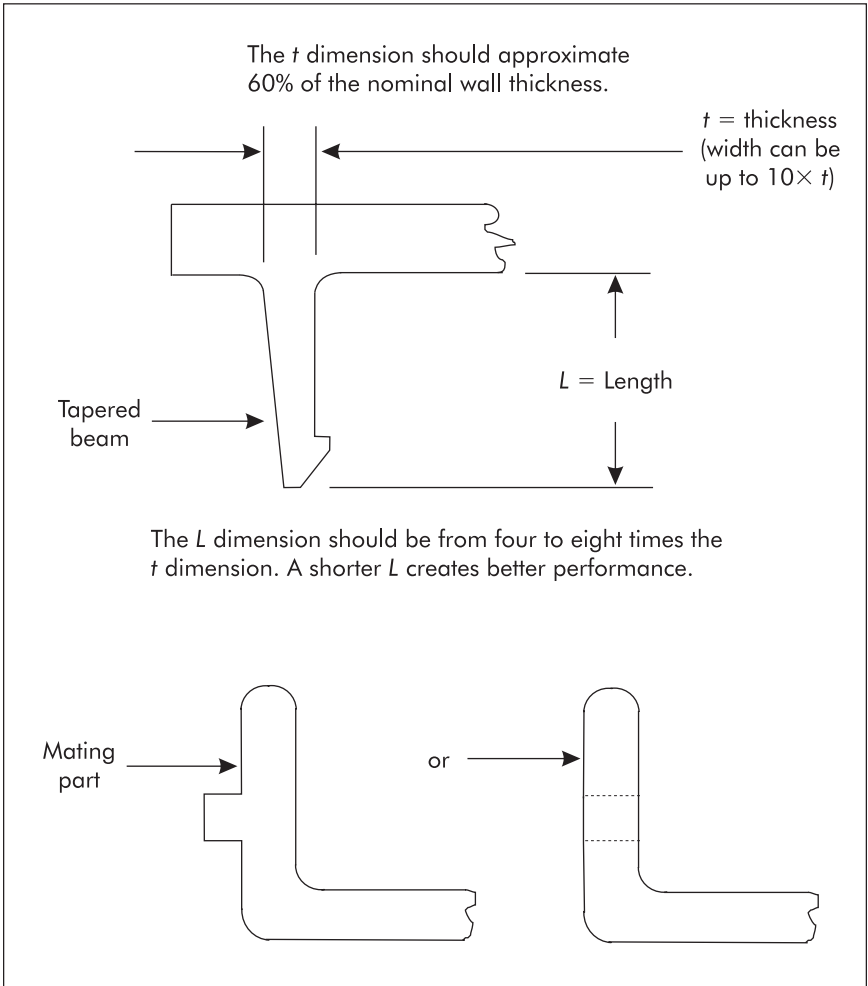


Figure 11-12. Common snap-fit (cantilever beam).

beam will result in 30% less generated stress than a straight-beam design. Also note that there are radii at the connecting points of the latching arm to the main wall of the plastic part. This greatly improves the ability of the arm to withstand flexing stress and, again, distributes the stress evenly over the length of the beam. The corner radius should be a minimum of 25% of the nominal wall thickness.

The most important factor to consider in designing a snap-fitting latch is the ratio of the length of the latch to its thickness. Contrary to popular opinion, a latch with a low length to thickness ratio (l:t) will sustain less material damage in operation, and thus will outperform a latch with a high l:t ratio. The longer latch tends to create greater “residual deflection.” Residual deflection is the percentage of initial deflection lost after deflecting the beam. It reflects the amount of material damage that occurred during that initial deflection. If there was no damage to the material, there would be no residual deflection. Thus, the design limits vary depending on the latch dimension l:t ratio.

To minimize vibration, some designs require that the snap-fit latch be designed in a “preload” condition. This means that the latch is designed such that it does not return to its original position, but instead rests tightly against the mating part. This will result in stress relaxation over time, and reduce the amount of flexibility present in the beam. It is better to rely on the normal mechanical retention forces of the snap design than to force the latch to remain partially deflected to gain retention force.

Another important factor in snap-fit design consideration is the plastic material being used. Materials perform differently, some withstanding higher levels of induced strain than others. The designer must work with the material supplier to understand the performance characteristics and limitations of a specific plastic. Pure cantilever-beam analysis alone is not adequate for proper snap-fit latch design.

Other Assembly Methods

Ultrasonic welding. The most popular method of assembling plastic parts today is the sonic vibration process.

The ultrasonic assembly process is detailed in Chapter 8. Here we examine some of the design parameters.

In this process, a metallic “horn” focuses sonic energy from a generator (Figure 11-13a) to the plastic parts being assembled. For parts made of amorphous materials, welding is effected by a designed-in energy director which concentrates the ultrasonic energy in a small area, shown in Figure 11-13b as the protrusion on the upper part.

When energized, the horn causes a vibration in the energy director, and the vibration results in friction. The friction results in heat which melts together



Figure 11-13a. Ultrasonic assembly machine. (Courtesy Branson Ultrasonics Corp.)

the surrounding plastic areas in a strong weld. There are many ways to design the energy director, but the concept is always the same: to focus ultrasonic energy in a small area. If the energy director can be designed to encircle an entire part, a hermetic seal can be developed.

When welding semicrystalline, or crystalline materials, the plastic parts should be designed to cause a slight interference fit between them as shown in Figure 11-13c.

This design is known as a “shear/interference joint” in which the sonic vibrations from the horn are dissipated through both plastic parts, causing melting where they meet. This melting continues along the vertical wall as the parts telescope together (from light, applied pressure), creating a continuous joint. This joint is usually stronger than the original parts being joined, and can be leakproof if designed to continue all the way around the parts.

Staking. The staking process may be used when a permanent attachment of two or more parts is desired. The process consists of applying either a heated tool or ultrasonic horn against a protrusion on the plastic part, with enough force to cause the protrusion to

flatten out, much like flaring the end of a rivet (see Figure 11-14).

With ultrasonic staking, the sonic energy causes vibration, friction, and subsequent heating of the plastic protrusion, which becomes formable and will flatten with slight applied pressure. When using heat staking, the heat is supplied by a heated tool. The tool is forced against the protrusion while the protrusion softens, resulting in a flattened stud.

Other basic assembly methods (such as solvent bonding) used to join plastic parts are discussed in Chapter 8.

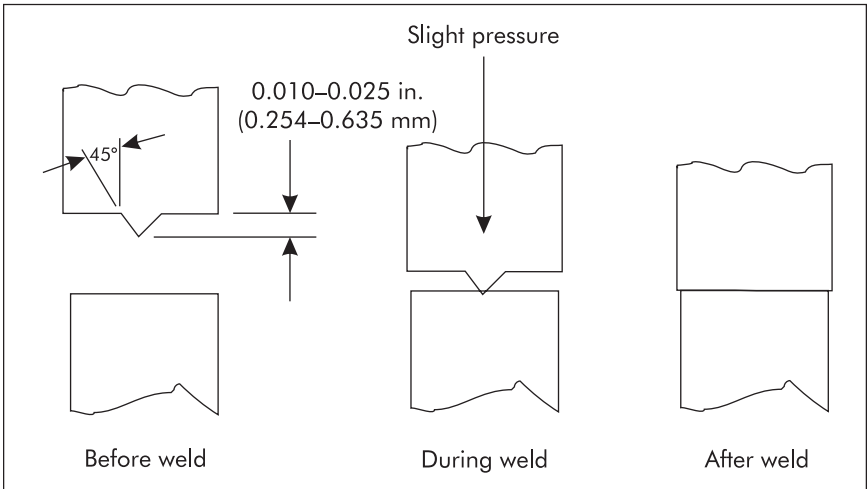


Figure 11-13b. Use of energy director.

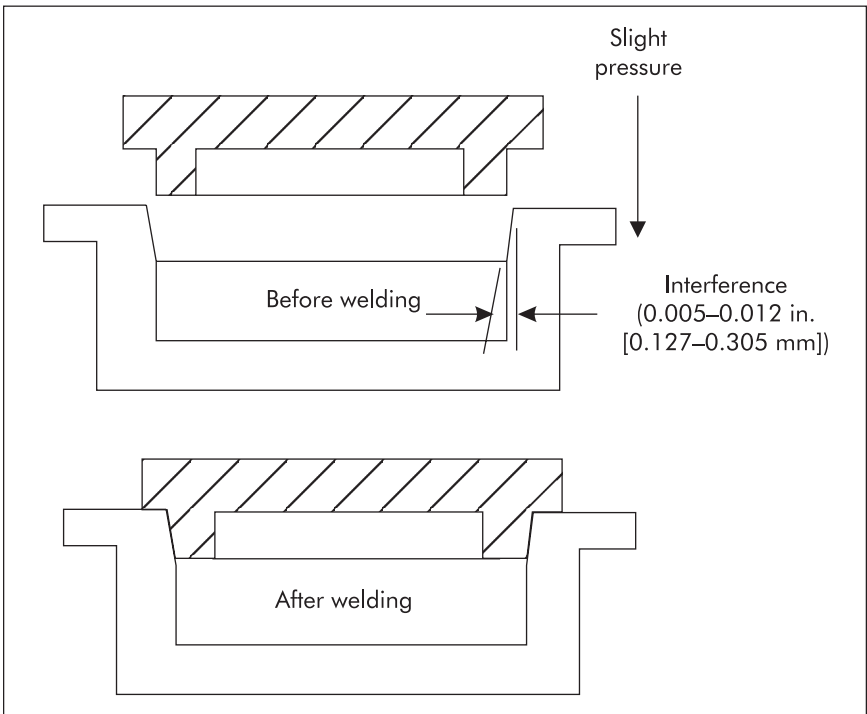


Figure 11-13c. Interference joint design.

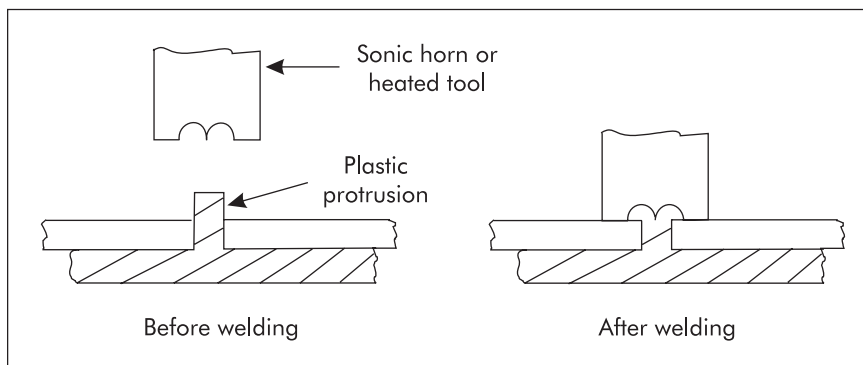


Figure 11-14. Staking process.

Thermal Aspects

Thermoplastic materials are thermal insulators. They do not readily absorb heat, but extended exposure to elevated temperatures will result in varying degrees of degradation. Each plastic has a thermal point beyond which permanent degradation occurs. One of the most common reference points is the *heat deflection temperature* (HDT). This is the temperature at which the plastic molecules are moving sufficiently to cause deformation of the plastic product. It varies with each plastic material, and the test for HDT is performed under a specific load. Chapter 9 details this test procedure. It should be noted here that the HDT should *not* be used to determine the end-use temperature value of a plastic product. The end-use temperature is the maximum temperature at which a product can be used without fear of failure or degradation. Both the actual HDT and the actual end-use temperatures are readily available for specific materials (and specific grades of those materials) from the material supplier's data sheets.

Color plays an important role in end-use temperature qualities. Dark colors will absorb heat while light colors tend to reflect heat. Therefore, products that are exposed to direct sunlight should be neutral or light in color. Outdoor tests in direct sunlight have shown that a black ABS panel with a foam backing can reach temperatures upwards of 175° F (79° C) compared to the same panel in white, without foam backing, which will reach only 125° F (52° C).

Crystalline materials tend to stay structurally sound when exposed to elevated temperatures because they hold their originally-molded properties until close to their HDT. Amorphous materials, on the other hand, tend to sag and weaken over a wider low-temperature range due to their random molecular structure.

Of course, exposure to below normal temperatures (normal being 73° F [23° C]) may also affect various properties of a plastic product. In the general area of freezing temperatures (32° F [0° C]), some plastics exhibit very little change

while others may exhibit drastic changes. These results will be noted in the material supplier's data sheets. Special attention should be paid to impact strengths at these low temperatures. Brittleness at low temperature is not easily predictable, but is undesirable for most products.

Electrical Aspects

Dielectric (insulating) properties of thermoplastics are exceptional and account for their wide use in electrical and electronic applications. A standard measure of conductivity is the electrical resistance of a material when a direct current is applied to it. This test is known as *volume resistivity* and is measured in ohms times the area of the smaller electrode divided by the thickness of the specimen being tested (see Figure 11-15).

Therefore, the value of volume resistivity is normally given in ohm-cm. Materials with values above 10^8 ohm-cm are considered to be insulators, while those between 10^3 and 10^8 ohm-cm are considered to be partial conductors. Table XI-2 shows the volume resistivity values for some commonly specified plastics.

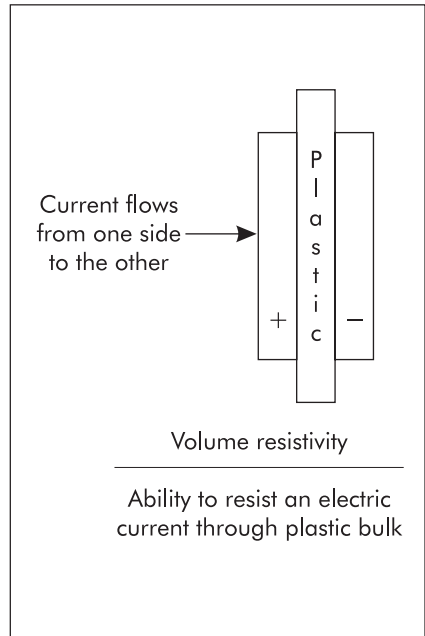


Figure 11-15. Volume resistivity test.

Flammability Considerations

While most products designed today must meet certain flammability requirements, there is some confusion as to how those requirements are determined. First, it should be noted that a flammability rating is *not* applied to the plastic part being molded, but rather to the material from which the part is molded. The test itself (UL regulated in the U.S.) is performed on specially-molded samples of the material being tested. These samples are conditioned and tested to rigorous standards, and a final value is given which can be cited by the designer (or other owner of a product design) as to the flammability rating itself. In effect then, the designer is free to state that the product “is molded of a material that meets UL flammability test so-and-so.”

The UL test is performed on a specimen of a specific thickness. The thickness corresponds to the UL rating being sought, and it should be noted that the thicker the specimen, the greater the flammability rating. Most molded plastic

Table XI-2. Typical Volume Resistivity Values

Material	Volume Resistivity (ohm-cm)
Acetal	10^{14} - 10^{16}
Acrylic	10^{16} - 10^{18}
ABS	10^{15} - 10^{17}
Nylon	10^{12} - 10^{16}
Polycarbonate	10^{15} - 10^{17}
TP polyester	10^{14} - 10^{17}
Polypropylene	10^{14} - 10^{17}
Polysulfone	10^{15} - 10^{17}
Modified PPO/PPE	10^{15} - 10^{17}
Polyphenylene sulfide (PPS)	10^{15} - 10^{17}
Polyarylate	10^{16} - 10^{17}
Liquid crystal polymer (LCP)	10^{14} - 10^{16}

product designs seek flammability ratings for specimen thickness values of 0.032 through 0.090 in. (0.813 through 2.286 mm). It is recommended that the designer become thoroughly familiar with the test requirements of whatever flammability rating is being considered.

Weather and Exposure Limits

Some products are intended for outdoor use, while others, intended primarily for indoor use, may be exposed to outdoor conditions at least some of the time. Still others are intended for use in situations which result in exposure to harsh conditions such as the floor of a manufacturing plant. In all of these cases the designer must be fully aware of the end-use *potential* exposure to the product, not just the primary *intended* exposure.

Weatherability testing results can be used to assist in selecting a material for typical outdoor applications. These tests evaluate the ability of a specific material to withstand exposure to ultraviolet light (from sunshine), exposure to varying levels of water and saltwater spray, and exposures to other materials and conditions such as sand and wind. These tests can be performed in actual, real-time situations such as at the beach in Florida, or in accelerated chambers in a laboratory.

Care should be taken when selecting materials for products exposed to water or moisture. Direct exposure may cause eventual permanent swelling and subsequent dimensional changes. Intermittent exposure may cause temporary swelling. Hygroscopic materials such as nylon and ABS will absorb moisture from the surrounding atmosphere and increase in size. Data available from the specific material supplier address such characteristics as weatherability.

Exposure to solvents, acids, and other chemicals may have degrading effects on some plastics. While the crystalline materials are much more resistant to chemical attack, a thorough understanding of chemical exposure (and potential exposure) to a particular product design is highly recommended before settling on a specific material selection. Working with various material suppliers and computerized data bases will assist in that final selection.

Safety and Environmental Issues

Of course, any product design must include a full understanding of the safety issues associated with it. Sharp external corners, thin rigid webs, and use of brittle materials are all examples of dangerous product attributes that can result in unnecessary end-user risk and injury. But in addition to these obvious items, the designer should think ahead and consider such things as where the product likely will be stored before and after purchase. And how will the product be disposed of at the end of its life? Also, has the product been designed to encourage recycling? These concerns all have eventual effects on safety and environmental issues.

Recycling efforts should be a major consideration when designing a product. If plating or painting processes are used, the result will be a plastic that cannot easily be recycled because of inherent dangers in the materials used for the plating or painting. Could the color be molded in instead? Could the plating be replaced with a removable metallic film? Can screws for assembly be replaced with snap-fit devices designed as integral parts of the plastic product? Addressing such issues will result in products that are more suited for recycling.

To assist in recycling efforts, the Society of the Plastics Industry has designed a marking strategy that readily identifies the generic material from which a product is made. This strategy is in constant flux and subject to change as time goes by, but at present consists of placing a chasing arrow symbol, Figure 11-16, (borrowed from the paper

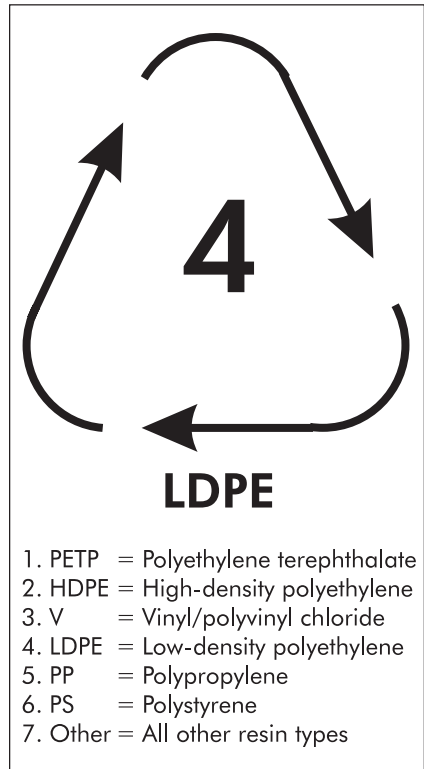


Figure 11-16. Recycling symbol and chart identifying material types.

industry) on the surface of the molded part. This symbol also contains a number from 1 to 7, and has the acronym for the generic material being used.

In Figure 11-16, the number 4 shows that the plastic is a low-density polyethylene, which is confirmed by the acronym LDPE shown below the number. Although this system is fairly specific and only identifies generic materials, it does cover the majority of consumer products made today. These include milk jugs, plasticware, toys, and household products. The number 7 materials, identified only as “other” are not used in nearly as high a volume (individually) as the 1 through 6 materials. For now, this system works well in helping recycle the most commonly disposed-of plastics.

A final consideration concerning safety and environmental issues: most plastics give off toxic fumes when they are burned. If an end-user accidentally or intentionally burns a plastic product, that plastic will undoubtedly release toxic fumes of some type. For instance, acetal material gives off formaldehyde gas, and vinyl plastics give off hydrochloric acid and chlorine gas. These are dangerous gases and chemicals and can cause severe physical damage. Care should be taken to discourage accidental or intentional burning (with warning labels, flame retardants, etc.). While there is no doubt a high degree of responsibility rests on the shoulders of the end user, it is the sign of being a good corporate citizen for the product manufacturer and designer to take appropriate steps to avert potential disaster.

SUMMARY

A product should be properly defined early in the development process to ensure the correct selection of materials, processes, and testing procedures. This will avoid constant engineering changes (and subsequent cost overruns and time delays).

In defining a product, the designer is usually the one with total responsibility. However, the successful designer will utilize every tool possible to create the proper definition. That includes materials engineers (not just materials selection data bases), tool engineers, process engineers, manufacturing engineers, quality assurance personnel, purchasing personnel, design guides (available from materials suppliers), and consultants, when necessary. It also includes tapping into the expertise of any vendors selected to work on the program.

There are seven basic areas of interest to consider when determining the actual requirements of a product to be manufactured. These are functional, physical and mechanical, thermal, and electrical aspects; flammability rating; weather and exposure limits; and safety and environmental issues.

The product designer must be aware that some materials are capable of producing parts to very tight tolerances, while others are not. The key to specifying

proper tolerances is for the designer to request only the amount of tolerance needed for the specific application. The reason for this is that tighter tolerances cost more money to produce.

When considering using metal inserts instead of molded-in threads, the ultrasonic insertion process produces the greatest degree of retention, followed by the heat insertion method and the cold insertion method.

A “living hinge” (integral hinge) product can only be successful if the hinge is flexed once or twice immediately after being removed from the mold. This sets the orientation of molecules properly for a lifetime of up to 300,000 flexures.

QUESTIONS

1. What does “defining the product” include?
2. What are the seven basic areas of interest to consider when determining the actual requirements of a product to be manufactured?
3. What should be done to ensure accuracy when using a color chip as a color sample?
4. Why is it not possible to manufacture a product to the *exact* dimensions placed on a product drawing?
5. What happens to *all* materials (except water) when they are heated, and then when they are cooled?
6. Name the three basic shrink factor groups for thermoplastics and give the shrinkage range for each.
7. How can strength and support be added to a boss without increasing the wall thickness?
8. Why should a flat-head screw design not be considered for assembling plastic products?
9. What must be considered to ensure adequate screw (or bolt) retention?
10. When should threaded metal inserts be used?

Product Design Considerations

12

DESIGN GUIDELINES

Many concerns must be addressed during the design phase of a product. After making the initial determinations discussed in earlier chapters, the designer is ready to place the final concepts on paper or in a computer file. Illustrated in Figure 12-1 are some of the many design guidelines that should be followed to produce a good, manufacturable, cost-effective, plastic molded product.

This drawing shows some of the critical areas with which a designer needs to be concerned. All walls should be of equal thickness if possible, but if a thicker wall is needed, a gentle transition (shown in the upper right section) should be specified. Ribs should be $\frac{2}{3}$ the wall thickness (to avoid sink marks), but gussets should only be $\frac{1}{2}$ the wall thickness. Sharp corners should be eliminated by using radii, and bosses have a direct dimensional relationship to the main inside diameter.

Wall Thickness Recommendations

Properties of thermoplastic materials enable them to flow when they are heated to the proper temperature (which varies with each plastic). This ability can be tested for, using a melt index unit per ASTM Test #D-1238. Ability to flow also determines how far a plastic can be injected through a specific wall thickness of a product. If it is an easy-flow material (with a high melt index), it can flow much farther than one that is not. In addition, the easy-flow material can flow through a much thinner wall section than one that is not. Low-flow, “stiffer” materials tend to cool and solidify before they can flow as far as easy-flow (less stiff) materials. It can be seen, then, that some materials can fill thinner walls easier than others, depending on the material’s melt index value or stiffness.

Table XII-1 is a listing of common materials and the wall thicknesses they can flow through.

This table shows, for example, that crystalline nylon (most common grade) is an easy-flow material because it can be injected through a minimum wall thickness of 0.015 in. (0.381 mm), while polycarbonate is harder to fill and requires a minimum wall thickness of 0.040 in. (1.016 mm). Note also that the maximum wall thickness for nylon is 0.125 in. (3.175 mm), while that for polycarbonate

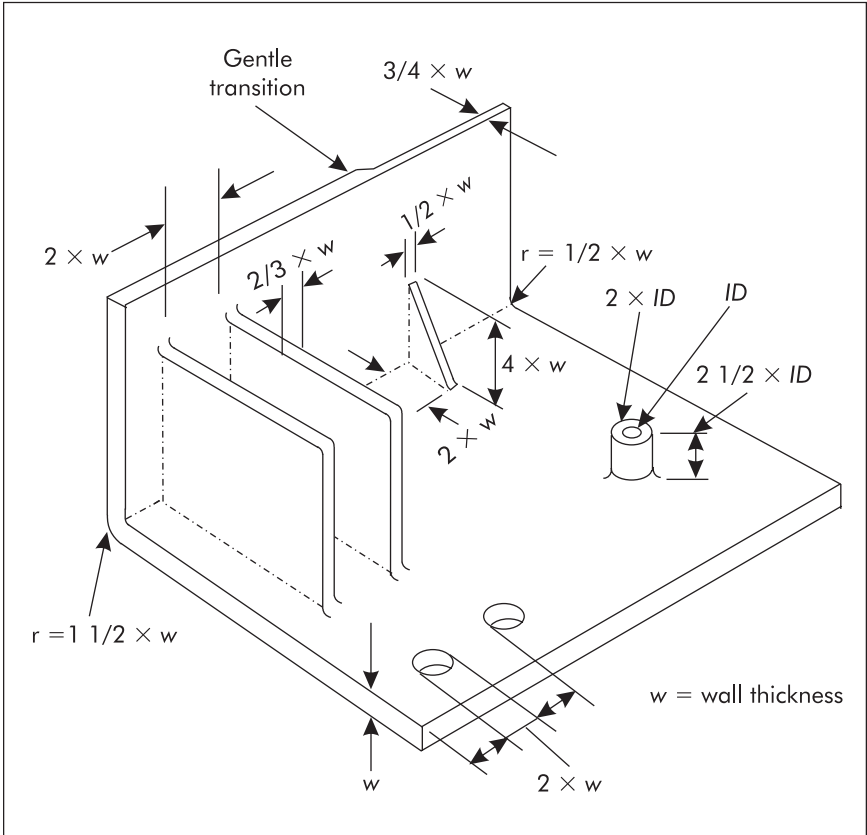


Figure 12-1. General design considerations.

is 0.400 in. (10.160 mm). These numbers show the thickest wall that can be molded without excessive warpage and twisting of the finished product. But the thicker sections also add to overall cycle times because it takes longer for these sections to solidify than the thinner sections. A wall thickness greater than 0.080 to 0.100 in. (2.032 to 2.540 mm) (depending on the material) will result in cycle time increases of 50 to 100%. Remember that heated plastic will solidify from the outside toward the inside and, therefore, owing to its great insulation properties, thick plastic material may take up to 30 days to finish cooling. The thinner the wall thickness, the faster this cooling and shrinking occurs. The cooling time can be a concern when the molder expects the plastic part to attain and hold accurate dimensions. If the part is dimensionally inspected too soon, inaccurate measurements may result. Because the greatest amount of shrinkage actually takes place within the first hour after part ejection for *average* wall

Table XII-1. Typical Wall Thickness Ranges for Common Materials

Material	Minimum (in./mm)	Maximum (in./mm)
ABS	0.030/0.762	0.125/3.175
Acetal	0.015/0.381	0.125/3.175
Acrylic	0.025/0.635	0.250/6.350
Nylon (amorphous)	0.030/0.762	0.125/3.175
Nylon (crystalline)	0.015/0.381	0.125/3.175
Phenolic	0.045/1.143	1.000/25.400
Polycarbonate	0.040/1.016	0.400/10.160
Polyester (TP)	0.025/0.635	0.125/3.175
Polyester (TS)	0.040/1.016	0.500/12.700
Polyethylene (HD)	0.020/0.508	0.250/6.350
Polyethylene (LD)	0.030/0.762	0.250/6.350
Polypropylene	0.025/0.635	0.300/7.620
PPO (modified)	0.030/0.762	0.400/10.160
Polystyrene	0.030/0.762	0.250/6.350
PVC	0.040/1.016	0.400/10.160

Note: These values are for rigid and semirigid materials. Values for foamed and flexible materials will vary and are available from the material supplier.

thicknesses, the part can be inspected after that hour with reasonable assurance that further shrinkage will be minuscule and not measurably affect dimensions.

While the wall thickness recommendations are indicators, it is always best to design a plastic part as *thin as possible*, but as *thick as necessary*. Most designers tend to “overkill” and make plastic parts much thicker than needed. This results in excessive and costly cycle times and more difficulty in controlling dimensions. If thick areas are required for reasons of strength, it is better to use a thin wall and add strengthening ribs, as shown in Figure 12-1.

Uniform Wall Thickness Requirement

Molten plastic material flows through the path of least resistance. If all the walls of a specific product are equal, the plastic flows with little effort and little loss of pressure as it travels through the described flow path. All the molecules within the material will become equal in size and take up equal amounts of volume, with equal amounts of space between them, as shown in Figure 12-2a.

If the walls are not uniform, as shown in Figure 12-2b, the molecules are forced to take up the space available to them. They will be compressed while traveling through the thinner areas and expanded while traveling through the opened areas.

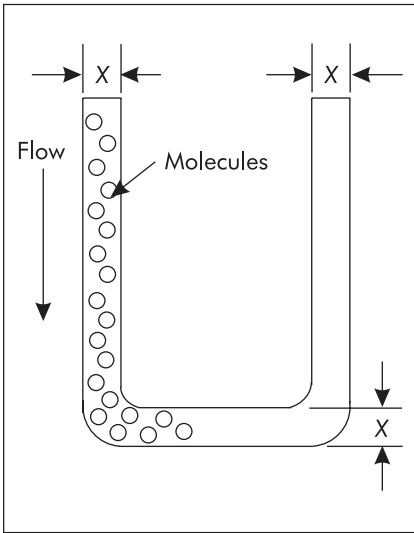


Figure 12-2a. Uniform wall thickness enables plastic molecules to flow without resistance.

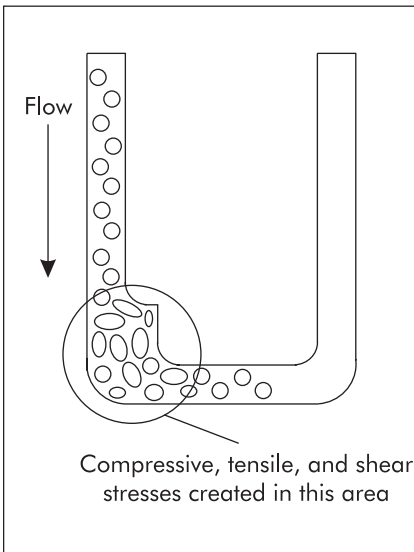


Figure 12-2b. Nonuniform wall thickness forces compression of plastic molecules, which results in stress.

This compression and expansion will result in compressive and tensile stress being created in the molten plastic molecules. The stress, in turn, will result in cracks, crazing, or fractures of the molded part at some point after molding when the stresses are released through mechanical, chemical, or thermal means. The failure may not occur for several years, or it can happen just a few minutes after molding. The stress will be released through possible chemical exposures, thermal attack if the product is exposed to temperatures at the low or high end limit for that specific material, or mechanically by the product being impacted in some manner. The physical properties of the molded product are greatly reduced through buildups of stress that occur as a result of nonuniform wall thicknesses. If the walls must change, stress can be minimized by designing in a gradual transition, such as that shown in Figure 12-2c.

If the part is designed such that the wall thickness increases gradually over the span indicated (rather than at direct right angles) a gentle expansion stress of the molecules results, with no shear stress. The molded product will maintain a greater degree of physical strength than one with an abrupt right-angle transition.

Knit Lines

Knit lines (or weld lines as they are sometimes called) are the result of a flow front of molten plastic coming in contact with an obstruction in the mold, breaking into two fronts to flow around

the obstruction, and meeting on the other side in an attempt to weld back together again. The faces of these fronts will cool somewhat while going around the obstruction and are not capable of welding back together to a 100% level. Thus, the fronts form what is called a knit line, which can be thought of as the starting point of a crack. The type of obstruction around which the plastic material might have to flow could be a core pin to form a molded-in hole, a molded-in insert, a dividing wall of steel forming a slot in the part, or any other such obstruction to the flow front. Figure 12-3a displays such a knit line as found when coring a molded-in hole.

The knit line will always form at a point opposite the gate, or opposite the area of impingement on the obstruction. If more than one gate is used, a knit line will form where the flow fronts of all gates meet, as shown in Figure 12-3b.

Once the conditions have been established to cause a knit line, the knit line cannot be eliminated. It can only be minimized (through processing and venting methods) or moved (by moving the gate or obstruction). Knit lines are usually the weakest area of a molded part.

Sharp Corners

A sharp corner on a molded part is an invitation to failure. Sharp corners are stress concentration points, developing stress in three basic forms: tensile, compression, and shear. Figure 12-4a depicts how these develop.

The circular objects in the walls of this cup-shaped part represent molecules as they flow through the wall section of the mold cavity forming the plastic part being molded. These molecules will take up the space available to them. As they enter the cavity they spread out and assume the circular shape shown. There are equal spaces between the molecules. As they enter the area of the sharp corner, some molecules become sheared, some must expand, and some are compressed as they again take up the space available while traveling through this area. It is here that stress develops. Any time a molecule is forced to take on a shape that it is not normally supposed to take, stress develops. That is what is happening in the area of the sharp corner, and that is why we consider the area a

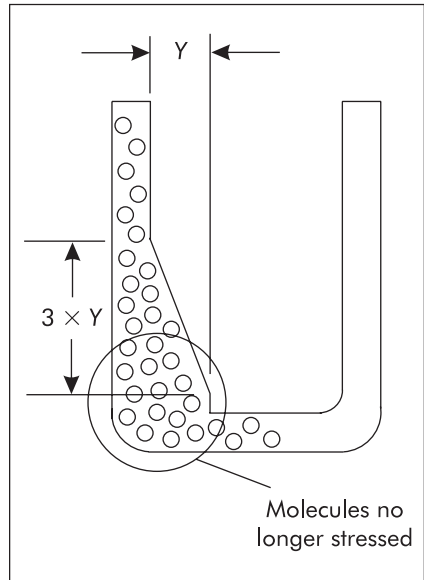


Figure 12-2c. Wall thickness transition.

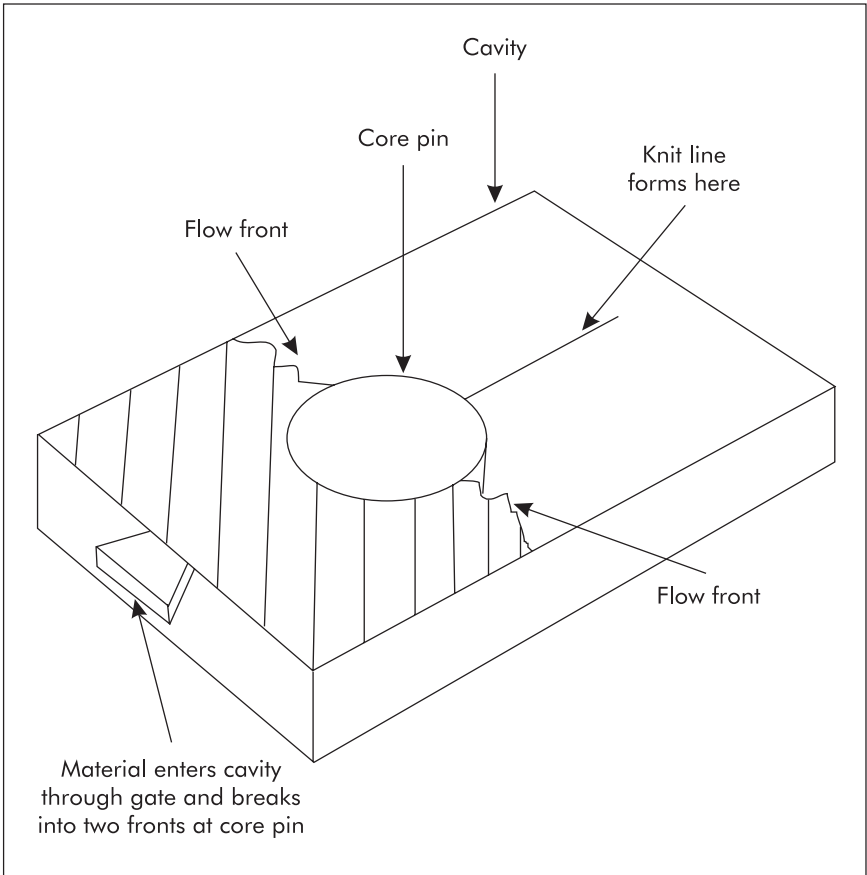


Figure 12-3a. Formation of a knit line.

stress concentrator. Note that the dimension across the diagonal of the corner is greater than the normal wall thickness dimension. This shows what happens as molecules are pushed through nonuniform walls. Again, stress develops. This area of stress concentration is vulnerable to release of stresses, making it extremely weak. Cracks or fractures are likely to occur.

Figure 12-4b shows how stress is eliminated (or at least greatly minimized) by adding radii to the otherwise sharp corner.

The inside radius can be as small as 0.015 in. (0.381 mm), and the outside radius should be a minimum of 0.125 in. (3.175 mm). Note that the molecules flow evenly through the walls, with equal spaces between them and equal amounts of open space. As they flow around the corner, they are kept in the same equal state and no stress is formed through shear, compression, or tension. The cavity

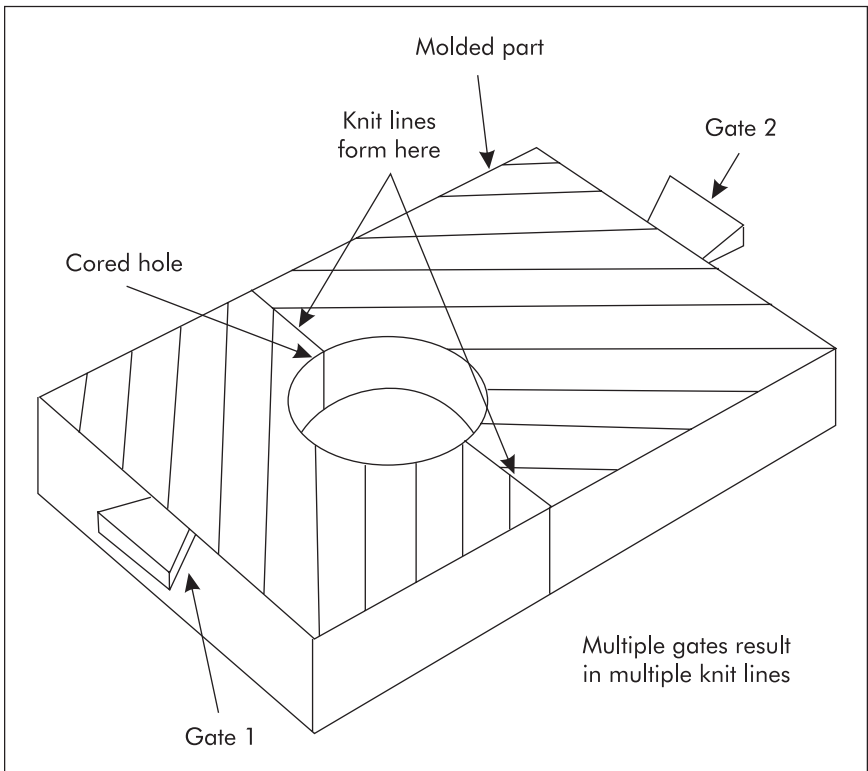


Figure 12-3b. Multiple-gate knit lines.

is filled and packed as the molecules and the space between them become more dense. This part will not crack or fracture due to stress concentration.

Sink Marks

Sink marks are formed when two areas of a molded part cool and solidify in a manner different from each other. This occurs when a thin section becomes solid sooner than a developed thicker section (Figure 12-5a).

Note that where the uniform walls meet, a section has developed that is thicker than the uniform walls. This thicker section will still be cooling (and therefore shrinking) well after the uniform walls have solidified and stopped shrinking. The material surrounding the thicker section will shrink in on itself and create a typical sink appearance. If this is in the area of a boss, the sink will be circular. If it is in the area of connecting walls, the sink will appear as a trench.

To keep the sinks from being apparent, make the connecting wall slightly thinner than the nominal wall as shown in Figure 12-5b. A good practice is to

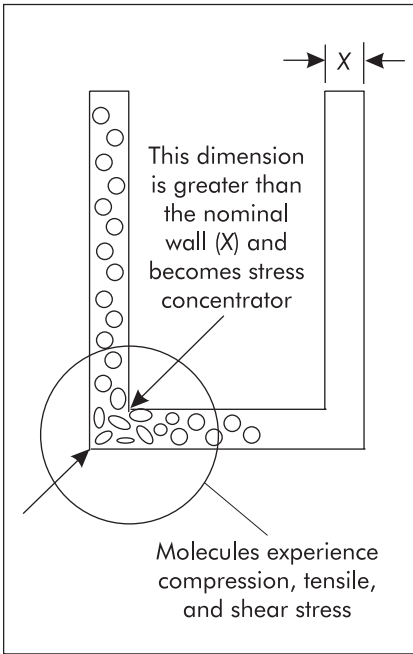


Figure 12-4a. Stress concentration points resulting from sharp-corner design.

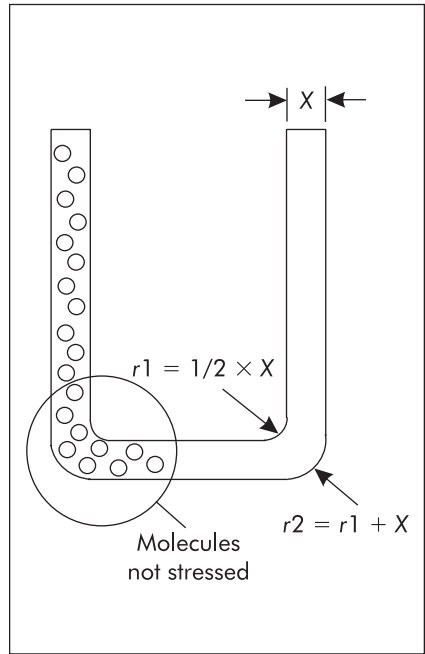


Figure 12-4b. Proper use of radii in sharp corners.

design the connecting wall to be between 60 and 80% of the thickness of the nominal wall. For instance, if the nominal wall is 0.090 in. (2.286 mm) thick, the connecting wall should be between 0.054 in. (1.372 mm) (60% of 0.090) and 0.072 in. (1.829 mm) (80% of 0.090). This results in a more uniform section in the connecting wall area which will cool and shrink at the same approximate rate as the surrounding wall sections. No sink marks will occur.

Warpage and Bowing

A molded part that appears warped, twisted, or bowed, is the result of stresses being created during the molding process or shortly after the time the part is ejected from the mold. This also can occur if the part is packaged while still too hot. Or it can happen if there is a burr on the mold steel that causes the part to get hung up during ejection and “snap” out of the mold while it is still hot enough to deform. It also can be due to improperly located gates in the mold or improper mold temperature control. Most common, however, is warping that can be traced to uneven or nonuniform wall thicknesses designed into the product. These nonuniform areas result in nonuniform cooling and therefore nonuniform shrinking. Thinner walls become solid while thicker walls are still shrinking.

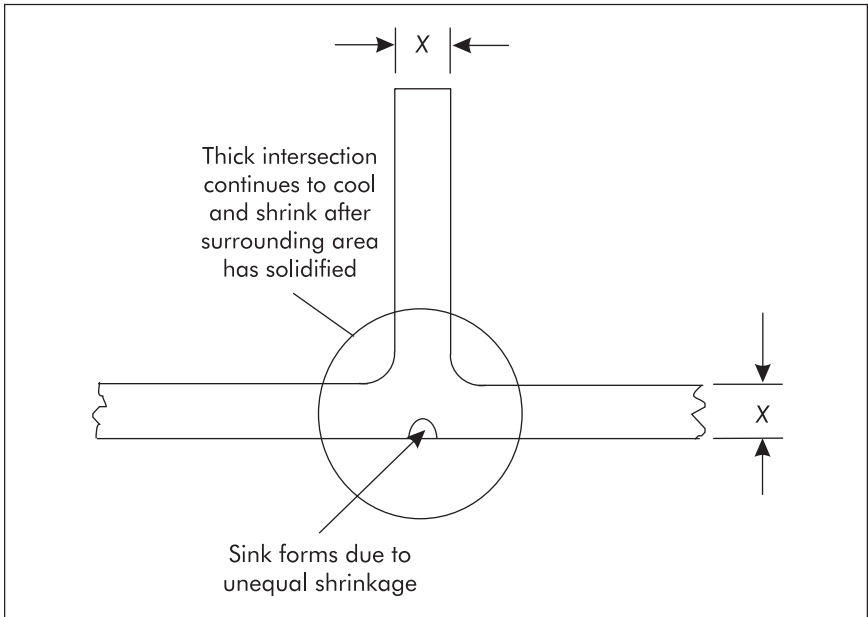


Figure 12-5a. Sink mark caused by uniform wall thickness design.

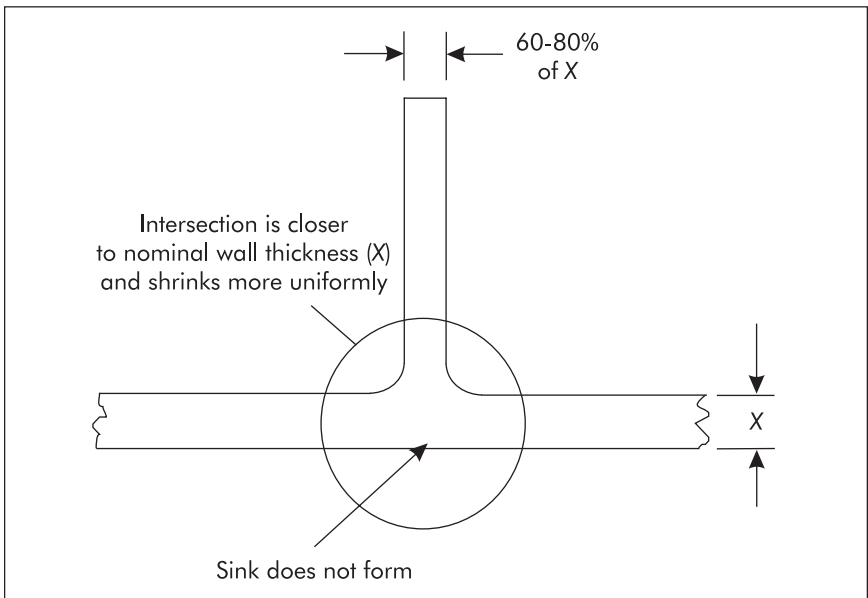


Figure 12-5b. Remedy for sink mark formation is a design calling for different wall thicknesses.

The thicker, still shrinking areas pull material in on themselves while cooling, causing the basic shape of the part to follow these shrinking areas. This results in high levels of stress which cause twisting, bowing, and warping. The use of uniform wall thicknesses, radii in sharp corners, and reduced rib and boss thickness will substantially eliminate the stress causing these conditions.

Draft

One of the most controversial elements of injection-molded product design is the requirement for draft. In its simplest terms, draft is defined as a taper applied to side walls (see Figure 12-6). It can best be illustrated by picturing an ice cube tray. Each of the tray cells has tapered walls to allow the cubes to eject easily. The same is true with injection molding: the molded product should have tapered walls to allow the part to eject easily from the mold. The amount of taper used is called the “draft angle.”

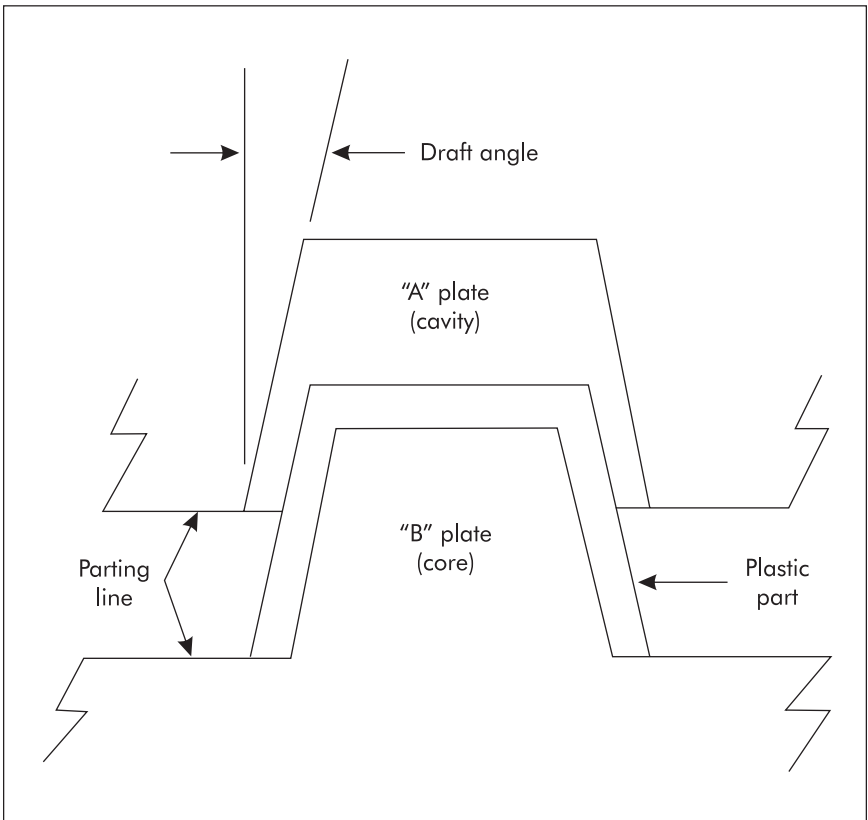


Figure 12-6. Draft angle defined.

The controversy surrounding draft concerns the amount of draft needed. The product designer would no doubt prefer *no* draft angle because it could have a negative effect on the function or visual values of the product. The molder, on the other hand, would like to see a great amount of draft because it makes it much easier to eject the part from the mold. The Society of the Plastics Industry (SPI) established some guidelines that help attenuate the controversy. The SPI states that injection-molded products should have a minimum of 1° of draft, 2° if possible. We examine, first, the effect of having no draft at all (see Figure 12-7a).

When a mold closes and is ready to accept incoming plastic material, air is trapped inside of the mold. This air will be displaced by the injected plastic. When the air is displaced, a vacuum occurs which will hold the part in the mold until air is allowed to enter. In a no-draft situation, the part must be ejected all the way out of the mold cavity before air can enter and replace the vacuum.

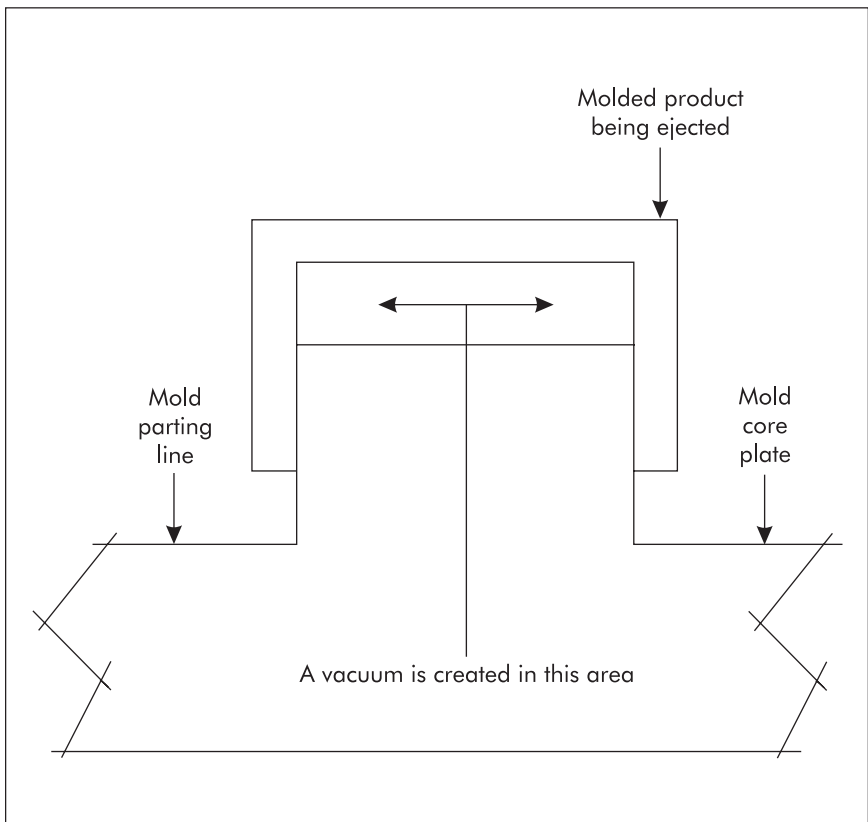


Figure 12-7a. With no draft angle, part ejection from the mold is very difficult.

Ejecting the part becomes extremely difficult. In fact, ejector pins may either break or puncture the molded part due to the excessive pressure required to eject the part against that vacuum. Figure 12-7b shows the effect of using 1° of draft.

The molded part has to travel only a fraction of an inch before air can enter the cavity and replace the vacuum. Once the vacuum is removed, the ejector pins can easily eject the part out of the cavity.

There is a price to pay for using draft angles, however. In Figure 12-7b we see that the part starts out with a 2.500-in. (63.500-mm) width. It also has a 2.000-in. (50.800-mm) height. The width dimension will grow when we add

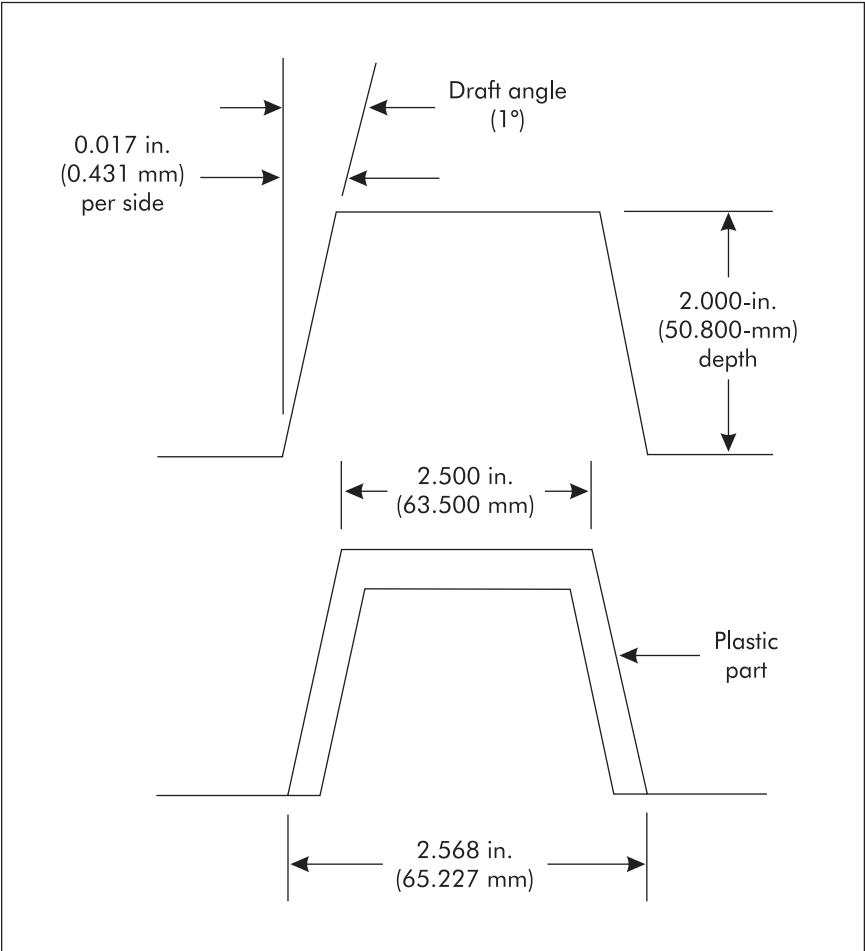


Figure 12-7b. Just 1° of draft permits sufficient air to enter the mold to enable easy ejection of the part.

draft to the side walls. For every 1° of draft we use, the dimension grows by 0.017 in. (0.432 mm) for each inch (mm) of depth, *per side wall*. In our case, the width will increase from 2.500 in. to 2.568 in. (65.227 mm). And if the side walls are textured, the amount of draft angle must increase by 1° per side for each 0.001 in. (0.0254 mm) of depth of texture used. Table XII-2 shows examples of the dimensional differences in inches (mm), per side, for some common draft angles at varying part depths.

Table XII-2. Examples of Dimensional Change versus Draft Angles (in./mm)

Depth	$1/2^\circ$	1°	2°	3°
1/25.4	0.009/0.229	0.017/0.432	0.035/0.889	0.052/1.321
2/50.8	0.017/0.432	0.035/0.889	0.070/1.778	0.105/2.667
3/76.2	0.026/0.660	0.052/1.321	0.105/2.667	0.157/3.988
4/101.6	0.035/0.889	0.070/1.778	0.140/3.556	0.210/5.334
5/127	0.044/1.118	0.088/2.235	0.175/4.445	0.262/6.655
6/152.4	0.052/1.321	0.105/2.667	0.209/5.309	0.314/7.976

While the use of draft is certainly desirable, it is possible to mold parts with no draft if the additional tooling costs can be accepted as well as the increased tooling turnaround time. A product can be designed in more than one piece, each piece designed without draft, and then all pieces assembled into one, as depicted in Figure 12-8.

Of course, using this method requires three (or more) individual molds and three (or more) individual molded parts, which then must be assembled to look like a single piece (if aesthetics are critical). Therefore, the tooling costs are greatly increased and part costs may become prohibitive, especially considering the labor-intensive assembly operations.

When using this concept it is possible to utilize specially-designed joints (where the three parts are assembled) to hide the assembly seams. These joint areas can be textured, depressed, or highlighted to look as though they were part of the original design concept.

Another way of producing no-draft parts is to use a single mold that incorporates devices known as “slides.” These devices are built into the mold and allow metal sections of the mold to move out of the way after the part has been molded and is ready to eject from the mold. Figure 12-9 shows an example of such a mold design.

In this design, the slides are machined to create the outer diameter of the molded part (a bobbin). They also form the top and bottom face of the part. The drawing shows the mold in the open position, after molding, and ready for ejecting.

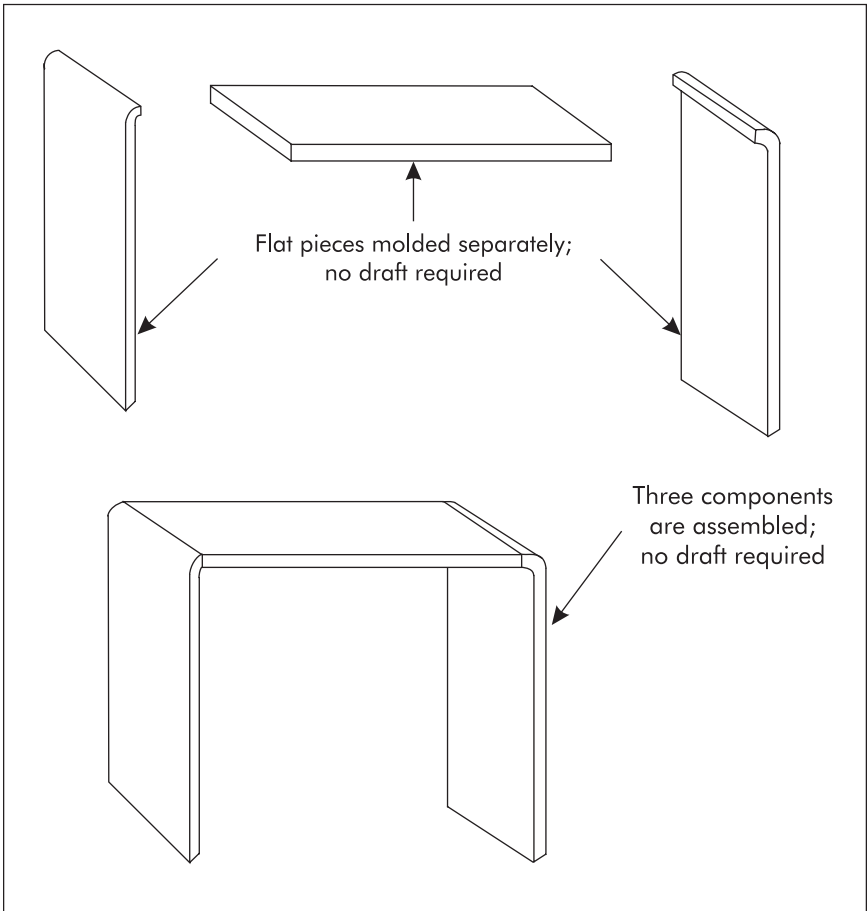


Figure 12-8. Designing with no draft.

When the mold is opened, the slides are pulled away from the plastic part by using stationary “angle pins” on the stationary mold half. These pins enter a slot in the slides and cause the slides to move backward when the moving half of the mold opens. The part is then ejected and the mold closes. The closing action of the mold causes the slides to move forward along the angle pins. There are other mold designs for no-draft parts, but they all are based on the concept of moving metal sections of the mold away from the plastic part so it can be ejected. These are expensive and can add anywhere from 40 to 400% to the cost of a standard mold that incorporates draft. And, because of the complexity of the mold, the time to design and build it may add 50 to 150% (or more) to the turnaround time.

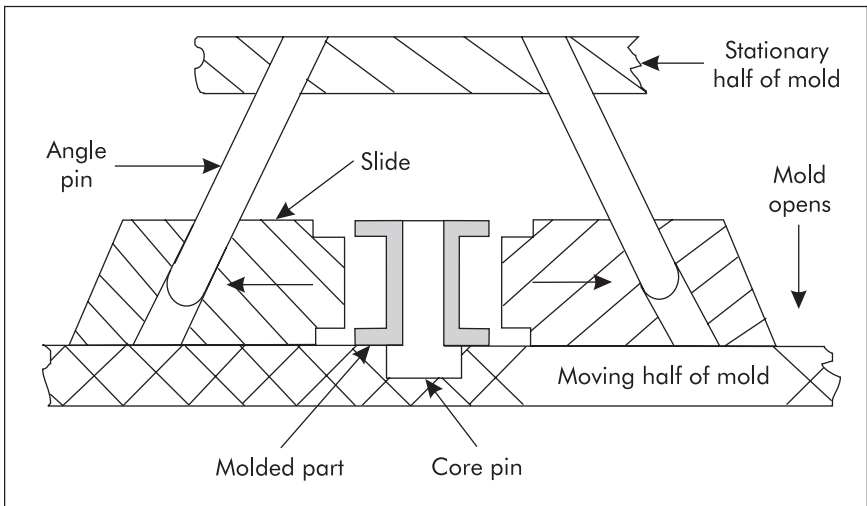


Figure 12-9. Special mold design for no-draft parts.

Ejector Pin Marks and Gate Location Restrictions

There are two areas of concern to which the product designer normally does not get sufficient exposure: ejector pin witness marks and gate protrusions or depressions.

Ejector pins are used for ejecting the part from the mold after the injection process is complete. Ejector pins actually form part of a face on the molded part. If the pins are made too short they will cause a “pad” of excess plastic material to form on this face. If the pins are too long, the plastic will flow around them and cause a depression in the part. Figure 12-10 depicts these conditions.

It is not possible to design an ejector pin having the exact length to form a flush surface on the part. Expansion and compression of the mold steel, tolerance buildups, moldmaking practices, and molding process parameters prevent it. Moldmakers intentionally accommodate for this by making the pins either long or short. It should be determined before (or during) the mold design stage which dimension the product designer prefers.

It is the product designer’s responsibility to indicate where *not* to have ejector pin witness marks, in case they interfere with aesthetics or the function of the part. Also, the designer should indicate, in areas where ejector pins might *be* allowed, how much pad or depression is acceptable. Normally, this is defined as a tolerance range of between 0.000 and 0.020 in. (0.000 and 0.508 mm). If no specification is made by the designer, the moldmaker and molder will use their best estimates and usually make the pins on the short side (causing a pad). This

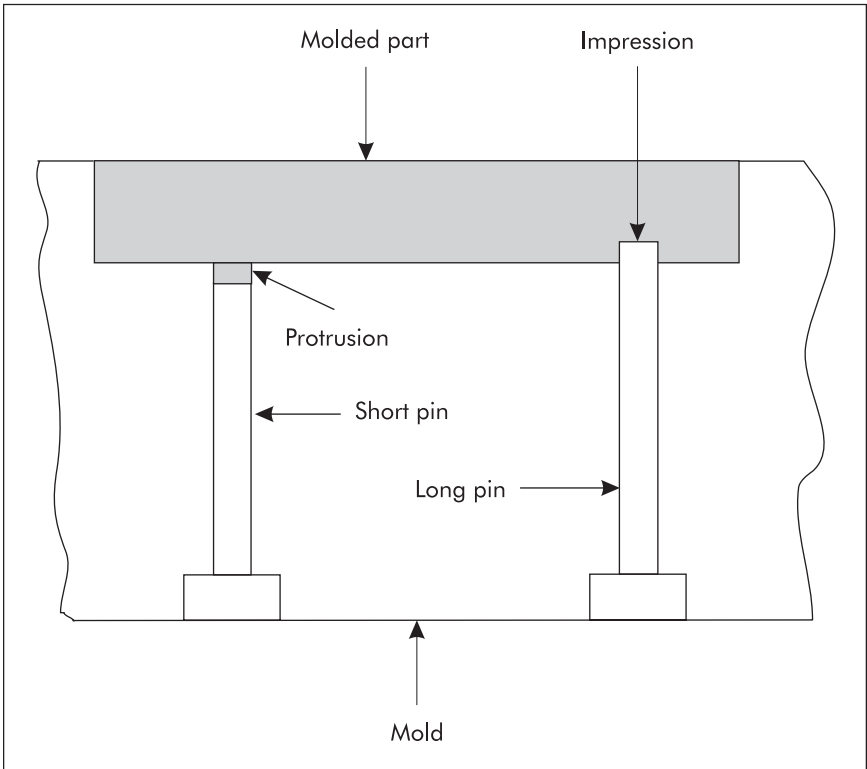


Figure 12-10. Ejector pin witness marks.

is preferred to having the pins long, which will result in the plastic shrinking around the exposed areas of the pins and hindering the ejection of the product from the mold. This condition should be avoided if possible.

Gate Marks

Gates are used to control the flow and volume of plastic entering the cavity of a mold. If surface gates are used, a depression or protrusion will be left when the molded part is separated from the gate, depending on how the gate was first designed (see Figure 12-11). If hot-runner or runnerless molds are used, there will still be a mark, but it will be less obvious.

Again, it is the responsibility of the product designer to indicate where *not* to have gate witness marks, if they might interfere with form, fit, function, or aesthetics of the finished product. Gate depressions or protrusions are difficult to control and may range within a tolerance of 0.000 to 0.125 in. (0.000 to 3.175 mm) or more. The moldmaker and molder will design, build, and locate

the gates according to the best design for controlling the process and molding the part. If the designer has any preferences for any reason, these should be stated and discussed before the mold is designed for build.

Any place on the mold where two pieces of steel come together to form a portion of the molded part will produce a witness mark of the steel pieces. Sometimes this is referred to as a parting line indicator, or simply a witness line. The designer should be made aware that such lines will occur, and he or she should work with the mold-maker or mold designer early in the mold design process (or even the product design process) to establish where they will occur. Most of these will not be detrimental to a product design, but some may result in visual marks that the designer would rather not have on the finished product. In such cases, the mold designer and moldmaker should work with the product designer to adjust the mold design or make recommendations to alter the product design.

Dimensions and Drawing Notes

Technically, the product designer's drawing takes precedence over all other methods of conveying the designer's wishes to the manufacturing side of the factory. Therefore, the drawings should be well thought out, and legible for all needing to view it. Standard formats should be followed, and dimensions should be inspected to make sure there are no cases of double-dimensioning, or dimensions omitted. These are both quite common and result in a great deal of confusion.

The dimensions of a drawing are what the mold designer, moldmaker, and molder look to for information regarding the complexity, function, and mechanical values of the molded product. In addition, there may be a series of notes including specific instructions regarding color, material, texture requirements, etc. that the designer wishes to convey to the molder. These are all helpful units of information if they are properly created and utilized. The product designer must take the responsibility for making sure the dimensions and notes are created properly, and the mold designer, moldmaker, and molder must take responsibility for making the product meet those dimensions and notes.

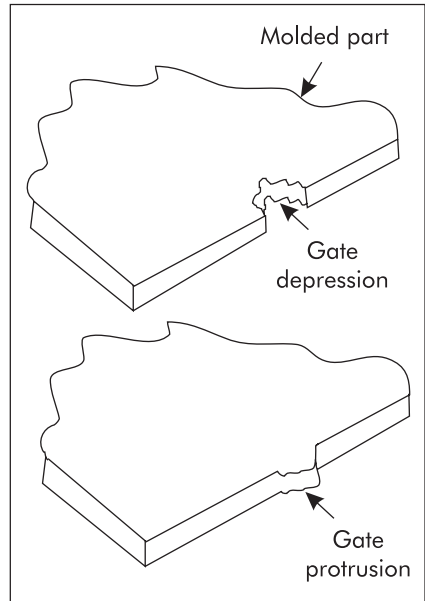


Figure 12-11. Gate witness marks.

For our purposes, it does not matter whether the dimensions are shown as metric, American English, or any other units of preference. What does matter is that it is explicitly indicated *which* units are being used. Never assume that the person reading the drawing will understand which units of measure are being used.

While there are many schools of thought as to what direction the numbers should run in relation to the drawing, it has been generally accepted that all dimensions should be placed such that they are read the same way, horizontally across the drawing. This keeps the reader from having to turn the drawing to read dimensions or from misinterpreting a dimension while attempting to read it sideways. There are usually more than 250 dimensions on an average product drawing. It helps to be able to easily see them all.

Notes on drawings are necessary evils. They are added in an effort to inform the manufacturing people of requirements outside the normal dimensioning information. While that is all right in itself, the average designer tends to place so many notes on a drawing that eventually the manufacturing people do not even bother reading them. If there are more than four notes on a drawing, the designer is not doing a proper job of conveying information through the basic product design. Items that *should* be in note form might include a specific plastic material to be used for molding (this usually also includes a color code), a flammability requirement (if applicable), indicators concerning ejector pin and gate marks (these also could be placed directly on the drawing), and a notice of any documents that are applicable to the specific product design (such as the customer's quality plan). Only notes that are specific to the particular product design should be on the drawing. Notes that are generic in nature (such as "A recycling symbol must be placed on the product") should *not* be placed on the drawing but instead in an overall design document referred to by the drawing. That way, the document can change without going through the hassle and expense of creating an engineering change procedure every time a simple note is reworded.

Remember that the drawing takes precedence over any other documents, but can *refer* to those documents. This means that to comply with drawing specifications, a vendor must read and comply with any document listed on the drawing. It also means that any other information not on the drawing or referred to by the drawing, such as a formal letter, scrap of paper, or a phone call, has no legal precedence over the drawing and *does not have to be acted upon*. This is why it is so important for the drawing to be correct and up-to-date.

WHAT THE VENDOR NEEDS

Product designers should try to put themselves in the vendor's place periodically. The vendor (whether it is the moldmaker or molder) is the vital link to

production from the designer's drawing input. If the designer designs a product such that it can be manufactured at the lowest cost with the highest level of quality, he or she will be rewarded with few engineering change requirements, faster turnaround times, and fewer headaches (and associated costs) in bringing the product to market. Following are some of the items the moldmaker and molder need from the product designer.

- *An accurate drawing.* This should be well-dimensioned, but not overdimensioned, and contain indicators for critical dimensions and areas. Cross-sectional views help the reader to visualize the concepts implied in the basic drawing design. Even when using computer-generated model designs (with no visible dimensions), accuracy is essential.
- *Adequate information.* Often, any notes that are used are copied from other drawings. Eventually, information gets dropped. For instance, when noting a flammability requirement, it is not enough to state that the product must meet a "UL flammability rating of V-1." The note must also state at what thickness this rating is expected, such as "at 1/16 in. (1.588 mm) thick." So be sure that adequate information is being supplied. Do not assume that another drawing has the correct notes or nomenclature. Check it out and be certain. Another common mistake is to list a material requirement in very basic terms when something greater is needed. For instance, the note, "Material to be glass-filled nylon" is not nearly specific enough because it allows any of approximately 200 different nylons to be used, and that does not include specific colors.
- *Complete materials information.* Someone must choose a material. The product designer has the final responsibility for this but should seek advice from a variety of others, such as the molder, material suppliers, material engineers, and consultants. In earlier chapters we discussed some of the areas to be considered. These include extreme operating temperatures, potential chemical exposures, operating environment, assembly processes, color and finish, cost per cubic inch, and availability. Discussing a few material selections with the molder will reveal processing influences that will facilitate selection of the most cost-effective material.
- *Critical dimensions or areas.* The more the vendors know about the function of the product(s), the more they can help point out potential hazards regarding product use, quality issues, cost concerns, and safety issues. With a full understanding of the critical requirements of the finished product, vendors will be able to select the best materials, processes, and tooling designs for the product.
- *Agency approval requirements.* More than a few products must meet various regulatory requirements, which usually mandate specific test results. Knowing those agencies which must be satisfied and what tests need to be

performed helps vendors determine procedures and related costs associated with meeting these standards. If this information comes only after releasing the purchase orders, one can expect major delays and excessive costs before the vendors are in a position to meet the needs. Of course, these delays and costs are passed on to the customer causing them.

- *Cooperation.* Vendor activities will come to an immediate halt if they cannot obtain required information in an expedient manner. Vendors should be given a single contact to call for any reason associated with the manufacturing of the designer's product. They should not have to wade through a series of purchasing agents, expeditors, engineers, and managers to get the information they need. A single person (and one backup) from the customer's company should be assigned the responsibility of working with the vendor to solve problems quickly or retrieve needed information. That individual must be willing to respond immediately and take on the task of locating the right people or information to help solve a problem. The more cooperation the customer can give the vendor, the less time will be lost bringing the product to fruition.
- *Accurate volume projections.* Even though they may only be best-guess estimates, the customer must be willing to give reasonably accurate annual volume requirements to the vendors. That is what vendors need to determine how many cavities to build in the mold, what size molding machine to use for producing the parts, and how to schedule to provide parts to the customer in a timely manner. Without these numbers the vendors will only be guessing at what the customer needs and this will greatly affect the product cost and delivery times.
- *Proper payment.* Usually a vendor will quote a project based on a deposit to accompany the purchase order and request periodic payments during the development stages of the project. The moldmaker will usually require a 50% deposit with the balance due upon acceptance of the first parts. Sometimes moldmakers use the 1/3 plan, which requires 1/3 down payment (with purchase order), 1/3 upon delivery of the mold, and the final 1/3 due upon acceptance of a molded part. Molders may ask for full payment in advance if the two have not partnered before. They may also ask for a deposit with the order (usually 50%) and the balance on delivery of the order. Whatever method is negotiated, the customer has the responsibility for, and should demonstrate ethical operating methods concerning, timely and proper payments. No vendor, of any size, can afford to carry debt for tooling, processes, or parts. It is neither fair nor ethical to withhold payments if negotiated terms have been satisfied.
- *Tooling approval signoff.* Unless otherwise specified, the customer owns the tooling. However, most customers do not supply a design to the moldmaker to build the tooling. The moldmaker normally designs a tool and sends a copy of that design to the customer for his or her perusal and

comments. If no comments are made within a specified time period, the moldmaker will proceed to make the tool. Once that is done, the molder (or in some cases the moldmaker) will sample the mold, make any necessary adjustments, and bring the mold to a production state. At that time the mold will produce what is usually referred to as “first-article” parts. These are given to the customer for approval. Once these are accepted by the customer, the tool is assumed to be accepted also, and tool payments are expected. This is usually done in a formal procedure, but even if a formal procedure does not exist, the customer does accept the mold by accepting first article parts.

- *Packaging requirements.* The vendor will assume bulk packaging is acceptable unless the designer specifically requests other packaging concepts, such as using dividers and separate protective bags or bubble wrap for each unit. If there is a potential that the parts may scratch each other, or otherwise become disfigured by bulk packaging practices, and it is important that they do not become so, the designer should note that special care must be taken in packaging to protect specific surfaces. It is also possible that the parts need to be packaged in special containers for future use in an automated assembly process. This is the type of information the vendor must have to provide an accurate packaging estimate. It also serves as an alert that special precautions may be required during the molding and parts-handling processes.
- *Shipment schedules.* The vendor needs to know how many units are expected each month, week, and quarter to enable him or her to schedule production runs that are the most cost-efficient for the vendor as well as the customer. Each time a mold is set in a press for production there are a number of fixed costs. If the production run is long, each part molded absorbs only a small fraction of those costs. If the run is short, each part absorbs a high percentage of those costs. The vendor would prefer to run the entire year’s worth of production in a relatively short time, usually over a 3-month period. The customer should give the vendor an estimate of the required shipping schedule early in the quoting process so the vendor is able to properly capture the associated setup costs and build them into the piece price.
- *Draft limitations.* Although the designer would like to have no draft at all, the requirement is there and must be satisfied. If there are areas where the designer wishes to hold critical dimensions, these should be noted so the vendors (moldmaker and molder) can take appropriate steps to ensure that the areas meet those demands. If desired, the designer can provide a specific note stating that a particular dimension must be held after applying draft by adding it to or subtracting it from the dimension. But usually it is better to simply note which dimensions must be held and let the vendors determine how to do it.

COMMON DESIGN CONSIDERATIONS

The following outline lists common design considerations that a product designer may wish to observe before finalizing a product design.

A. Basic design

1. Maximum function. Molds are expensive; the fewer individual components used in a product, the fewer number of molds will be required. Therefore, a product designer should strive to design into a single component as many functions as possible. Use of items such as snap fittings, threaded bosses, and supporting ribs not only help eliminate assembly operations, but also improve structural integrity and often reduce overall weight.
2. Optimal material. Several candidate materials should be selected based on product requirements such as operating environment, functional limitations, and end-user risk. Once several candidates have been selected, issues such as cost constraints can be considered.
3. Reduce material volume. Most designers specify excessive wall thickness, usually assuming that if a thin wall is good, a thicker one is better. This is not true, and will only result in higher costs and lower quality. The best choice is to use the minimum volume of plastic that satisfies the structural, functional, appearance, and moldability requirements of the specific application.
4. Exploit advantages. The injection molding process provides the means to accurately and repeatedly produce finished, multifunctional, or complex molded plastic products in a single, automated operation. With this in mind, consider the following questions:
 - a. Can more than one part be combined into a single part, thus eliminating extra materials, molds, molding operations, and assembly procedures?
 - b. Can hardware items or other components be eliminated by integrating them with the molded part?

If either question can be answered with a “yes,” further design analysis should be performed to take greater advantage of the injection molding process.

B. Material selection

1. Temperature. Mechanical and electrical properties, as well as chemical resistance properties, are dependent on environmental temperatures. A selected plastic must be capable of withstanding the expected normal and extreme operating temperatures of the product. Be sure to consider shipping and storage temperatures, potential

sources of internal heating, and temperature exposure during secondary, finishing, and testing processes.

2. Assembly operations. Certain materials are difficult to bond using solvents, while others do not lend themselves to ultrasonic methods of assembly. Be sure the selected material is suited for any intended assembly processes.
3. Environmental exposure. The selected plastic must be able to withstand exposure to every substance—solid, liquid, or gas—with which it can come in contact over its expected life span. The level of compatibility for all common substances is usually available from material suppliers' literature, and for those not listed, the supplier should be contacted.
4. Finishing. Color and texture can be provided most economically by the material and the mold. However, there may be a need for plating, painting, or other finishing processes as secondary operations. In either case, the selected material must be capable of providing the desired finish. For example, materials that are glossy by nature may not be capable of providing satin finishes. And some materials are attacked by the chemicals used in plating processes.
5. Approvals. The end product will probably need to meet certain agency approvals before being marketed. Underwriter's Laboratories (UL), the Canadian Standards Association (CSA), the U.S. Food and Drug Administration (FDA), and a host of other agencies, including a variety of military specifications must pass on the products before distribution. The ratings associated with many of these requirements are based on specific factors such as wall thickness, color additives, fillers, and operating temperatures. The material supplier will be able to provide compliance information on a specific type and grade of material.
6. Cost. While cost per pound of material used is important, a more meaningful number is the cost per cubic inch (or cubic centimeter) of material used. Of the 20,000 or more materials to choose from, some have high specific gravity levels (above 1.0 g/cm³ [0.04 oz/in.³]) and some have low specific gravity levels (less than 1.0 g/cm³). Therefore a cubic inch (cm³) of one may weigh considerably more than a cubic inch (cm³) of another. The formula for determining cost per cubic inch (cm³) is:

$$0.0361 \times \text{specific gravity} \times \text{cost per pound (kg) of material}$$

To determine a very approximate overall product cost, you can double the total cost of material.

7. Availability. Material suppliers tend to keep a limited supply of popular types and grades of material on hand, and usually must manufacture a specific requirement to order. It is common to find lead times for delivery of materials to be 4 weeks or longer. Also, small orders may take longer to place in the queue than large orders. Be aware of this and make an effort to select a material that is more common.

C. Nominal wall thickness

1. Structural integrity. While a wall section that is too thin can lead to poor performance or structural failure, a section that is too heavy can result in excessive weight, long (expensive) cycle times, or an unattractive product due to sink marks and other visual defects. Many aspects of the injection molding process are influenced by wall thickness dimensions, prompting the designer to be ever aware of the importance of using the correct nominal wall thickness. The majority of plastic parts molded today are within the nominal wall thickness range of 0.050 to 0.125 in. (1.270 to 3.175 mm). Some parts are being molded with wall sections as thin as 0.005 in. (0.127 mm) for short distances. And some parts are molded with wall sections greater than 0.125 in., but due to the almost exponential change in cooling time requirements for walls over 0.100 in. (2.540 mm) thick, it makes more economic sense to limit the maximum wall to 0.100 in. and add supporting ribs, bosses, or contours to increase a specific section modulus.
2. Insulation properties. Because plastics are such good insulators, it may be necessary to alter the wall thickness to attain specific resonant frequency for sound transmission, or light transmission of an optical product. In general, insulating properties are directly related to the thickness of the plastic.
3. Robustness. The impact resistance of a plastic product is dependent on wall thickness. Normally, increasing the wall thickness increases impact resistance. However, if the wall is too thick the part may become overly stiff, unable to properly distribute the impact, and thus become less impact-resistant. The properties of a specific material *and* the geometry of the particular product design must be analyzed when considering impact resistance.

D. Radii versus sharp corners

1. Molecular compression and stress. Sharp corners, particularly inside corners, result in poor material flow patterns, reduced mechanical

properties, increased tool wear, and severe molded-in stress caused by shrinkage variations in the corners. To avoid such problems, radii should be used in place of sharp corners.

2. Uniformity of flow. Note in Figure 12-12, that the inside radius is equal to half the nominal wall thickness, but never less than 0.015 in. (1/64 in., 0.381 mm). The out-side radius is equal to the inside radius plus the wall thickness. This results in uniform wall thickness and shrinkage through the corner section of the product. Uniform flow and uniform shrinkage result in no stress.

E. Undercuts

If undercuts are present in the product design, the tooling becomes much more complex and expensive and more prone to damage, resulting in excessive maintenance costs. Some unique designs are possible that will result in accomplishing the function required by the normal undercut situation, having the capability of being placed in the tool at less cost and with less potential maintenance.

The moldmaker and mold designer are good sources of assistance and should be involved as early in the product design stage as possible.

PRODUCT DESIGN CHECKLIST

Figure 12-13 depicts a suggested checklist product designers can use to determine the level of manufacturing acceptance of their product design. It is a common format, one that can be used for most product designs, but should be considered a fundamental form. Additions and deletions can be easily made to tailor the form to a specific product or company.

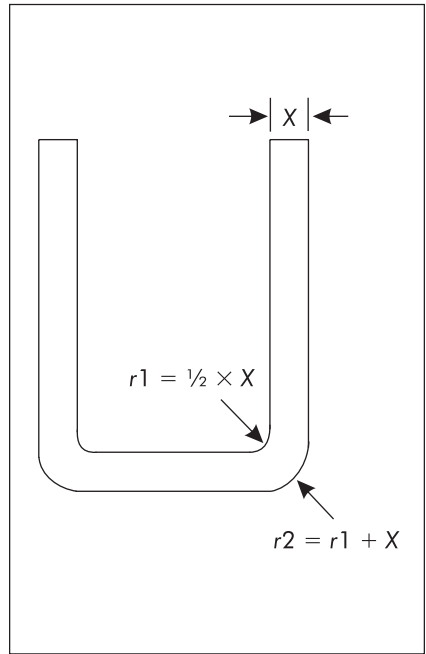


Figure 12-12. Radii versus sharp corners. Uniform wall thickness throughout the corners ensures uniform shrinkage, thus eliminating stress.

Plastic Product Design Checklist**Material**

- ☐ Availability.
- ☐ Correct code, type, grade.
- ☐ World trade alternates.
- ☐ Temperature (end use, shipping, storing, secondaries, etc.).
- ☐ Compatibility with assembly operations.
- ☐ Environmental exposures (chemicals, etc.).
- ☐ Compatibility with finishing processes.
- ☐ Agency approval status.
- ☐ Cost per in.³ (cm³).

Color

- ☐ Color approved?
- ☐ Color chip availability.
- ☐ UV stability.
- ☐ Clarity or transparency.
- ☐ Gloss level.

Texture

- ☐ Texture identification.
- ☐ Sufficient draft (1° per side per 0.001 in. [0.0254 mm] of texture depth).
- ☐ Texture-free area at parting line (0.030 in. [0.762 mm]).

Dimensions

- ☐ Missing dimensions?
- ☐ Double dimensions?
- ☐ Proper datum lines.
- ☐ Can part be inspected as dimensioned?
- ☐ Adequate draft allowance.
- ☐ Proper draft direction.
- ☐ Reasonable tolerances.
- ☐ Check for interferences.

Gate(s)

- ☐ Gate location preferences or nonallowances.
- ☐ Gate impression or protrusion notations.

Parting line(s)

- ☐ Approximate path of parting line.
- ☐ Is parting line aesthetically acceptable?
- ☐ Are product features compatible with parting line?

Figure 12-13. Plastic product design checklist.

Wall sections

- ☐ Uniform wall thickness.
- ☐ Sharp corners replaced with proper radii.
- ☐ 3-to-1 transition zone for thicker wall sections.
- ☐ Nominal wall compatible with material selection.

Ribs/bosses

- ☐ Proper rib-to-wall ratio to avoid sink marks.
- ☐ Proper boss design and location for material and inserts.
- ☐ No sharp corners at base of ribs or bosses.

Ejector pins

- ☐ Examine design for use of ejector pins.
- ☐ Note areas where ejector pin marks are not allowed.

General

- ☐ Dimensional compatibility of mating parts.
- ☐ Clarity and content of notes.
- ☐ Notes apply only to this drawing.
- ☐ Part number, date, and recycle symbol on part.
- ☐ List of appropriate specifications.

Figure 12-13. (Continued)

SUMMARY

Thermoplastic materials flow when they are heated to the proper temperature (which varies with each plastic). This ability can be verified by test, using a melt index unit per ASTM Test #D-1238. This ability to flow also determines how far a plastic can be injected through a specific wall thickness of a product. If the material is an easy-flow material (with a high melt index), it can flow much farther than one that is not. In addition, the easy-flow material can flow through a much thinner wall section than a material that is not easy-flow. Low-flow, “stiffer” materials tend to cool down and solidify before they can flow as far as the easy-flow (less stiff) materials. It can be seen, then, that some materials can fill thinner walls easier than others, depending on their melt index value and stiffness.

While the wall thickness recommendations are indicators, it is always best to design a plastic part as *thin as possible*, but as *thick as necessary*. Most designers tend to “overkill” and make plastic parts much thicker than needed. This results in excessive and costly cycle times and difficulty in controlling dimensions. If thick areas are required for strength purposes, it is better to use a thin wall and add strengthening ribs.

Molten plastic material flows through the path of least resistance. If all the walls of a specific product are equal, the plastic flows with little effort and little loss of pressure as it travels through the prescribed flow path. All the molecules within the material will become equal in size and take up equal amounts of volume, with equal amounts of space between them.

Knit lines (or weld lines as they are sometimes called) are the result of a flow front of molten plastic coming in contact with an obstruction in the mold, breaking into two fronts to flow around the obstruction, and meeting on the other side in an attempt to weld back together again. The face of these fronts will have cooled down somewhat while going around the obstruction and are not capable of welding back together to a 100% level. Thus, they form what is called a knit line, which can be thought of as the starting point of a crack.

A sharp corner on a molded part is an invitation to failure. Sharp corners are points of stress concentration, developing stress in three basic forms: tensile, compression, and shear.

Sink marks are formed when two areas of a molded part cool and solidify differently from each other. This is caused by a thin section becoming solid sooner than a thicker section.

A molded part that appears warped, twisted, or bowed, is the result of stresses being created during the molding process, or shortly after the time the part is ejected from the mold.

One of the most controversial areas of injection-molding product design is the requirement for draft. In its simplest terms, draft can be defined as a taper applied to side walls. It can be thought of in terms of an ice cube tray. Each of the ice cube tray cells has tapered walls to allow the cubes to eject easily from the tray. The molded product should have tapered walls to allow the part to eject easily from the mold. The amount of taper used is called the “draft angle.”

Ejector pins actually form part of a face on the molded part. If the pins are made too short they will cause a “pad” of excess plastic material to form on this face. If the pins are too long, the plastic will flow around them and cause a depression to be formed in the part.

If surface gates are used, when the molded part is separated from the gate, a depression or protrusion will be left, depending on how the gate was designed in the first place. If hot runner or runnerless molds are used, there will still be a mark, but it will be less obvious.

It should be stated that, legally, the product drawing takes precedence over all other methods of conveying a product designer’s wishes to the manufacturing side of the industry. Therefore, it should be well thought out and legible for all needing to view it. Standard formats should be utilized, and dimensions should be inspected to make sure there are no cases of double-dimensioning, or dimensions left out. These are both common occurrences resulting in a great deal of confusion.

QUESTIONS

1. What is the name of the machine and the ASTM test number to determine the “ability to flow” of a specific plastic material?
2. What does the flow test determine?
3. Which material will flow easier, a crystalline nylon or an amorphous polycarbonate?
4. What is the result of designing a product with too thick a wall section?
5. In its simplest terms, define draft.
6. Why is draft required on a molded product?
7. How much will a 2° draft add to each side of a width dimension of a part that is 3 in. (76.2 mm) deep?
8. What happens to a plastic part if the mold’s ejector pins are too short or too long?
9. In conveying a product designer’s wishes to the manufacturing side of the industry, which document takes legal precedence over all others?
10. What type of notes should be placed on a product drawing?

Miscellaneous Determinations

13

ANNUAL VOLUMES

The main factor in determining how many cavities will be needed in a mold is the annual volume requirement. It is necessary to be as accurate as possible with this estimate because molds are expensive, and the more cavities required the more expensive the mold. There is a trade-off, however. The more cavities running at one time, the lower the cost per unit being molded. Thus, it is necessary to determine the most efficient combination of the cost per unit and the total cost of the mold.

Annual volume estimates are normally educated guesses based on marketing surveys, present growth rates of a specific industry or sector, estimated consumer demands, formal projections published by government groups, and other factors associated with a particular product design. These guesses are normally provided by a marketing group, either internal or as a contracted service. Volume estimates are usually trimmed in an effort to create a more realistic environment for determining actual manufacturing costs.

It is prudent to realize that, for a normal 5-year life span, product volume requirements ramp up from just a few in the earliest stages of production, to approximately 50% of estimated maximum requirements by the end of the first year. This will increase to 100% of estimated maximum by the end of the second year. The 100% trend will continue through the third year, but by the fourth year the requirements will drop to approximately 50%, and by the fifth year they will have fallen to 10% or less.

An important factor regarding this roller-coaster effect is that the mold and other tooling must be capable of sustaining production requirements at every stage, during every portion of the year. Some companies elect to begin production with a less expensive mold designed to produce only one part at a time, then switch to more expensive, multicavity molds in the high production years. While the first year's parts may cost more on a per-unit basis, the lower cost tooling may result in a lower total manufacturing cost for those few parts. And initial investments are much lower during the first stages, while marketing has a chance to revise estimates upward or downward before committing to larger, high-volume tooling costs. In fact, prototype tooling can be utilized to produce the first year's production if it is designed for it in the first place.

As stated, this scenario holds true for products with a 5-year life span. Products with life spans greater or smaller than 5 years have annual ramp-ups and ramp-downs different than those mentioned, and these can be interpolated accordingly.

MATERIAL HANDLING REQUIREMENTS

It is advisable for the product designer to be aware of how the finished product is handled from the point it is ejected from the mold, through secondary operations (if required), and even into packaging and shipping. Knowing this product flow may lead the designer to select another plastic material, one which is more suited to the specific handling procedures.

For instance, it is most cost-efficient to mold parts in an automatic machine mode in which an operator is not required. But, in this mode, the parts usually fall from the mold onto a conveyor belt or into a container. If the plastic that is selected cannot withstand the force of the drop, or has a soft surface that can be marred by the dropping action, the machine may have to be switched to a semiautomatic mode and an operator added to remove the parts from the mold by hand. This will increase the total cost of manufacturing and has the potential of causing a higher level of defects due to the added human intervention. A change in material might be warranted, but the designer must be made aware of this need, and the earlier in the development process the better.

For secondary operations, handling of the molded product may be necessary for such activities as fixturing, locating, turning, machining, and a variety of others. It is possible that some, or all, of these may be performed by automated equipment or robots. In those cases, especially, it is better to design the product to be easily manipulated. For instance, if screws are used for assembly, a robot has difficulty differentiating between long and short screws, or screws with various head or slot designs. It might be possible to redesign the product so that all screws are the same. Or, if a part is basically round, it may not be possible for automated equipment to consistently locate it properly for a printing operation that must be performed in a specific location. A flat also could be built in that would assist in this activity. By being aware of the requirements, a designer can be instrumental in eliminating these kinds of barriers to efficient manufacturing processes.

SECONDARY OPERATIONS

Any product designed for secondary operations should be analyzed to ensure the material and design selected meet the established requirements. For example, a product requiring assembly using ultrasonic welding will require the use of energy directors to properly focus the ultrasonic energy to the joint area. But the more area being used for joining, the longer the process will take. A careful study, assisted by ultrasonic equipment engineers, will help ensure proper energy

director design and location. Also, some materials are suited to ultrasonic welding, while others will not perform well, based on molecular structure. Again, the ultrasonic equipment engineers are a good source of unbiased information concerning this factor.

The success of decorating a surface finish of any plastic product depends on the plastic itself. Some plastics (especially polyolefins) are difficult to decorate and usually require some type of surface preparation to obtain adequate adhesion of the decoration. There is a big difference between pigments and dyes, and some plastics can be decorated with only one type, while others may not be able to accept either.

Not all plastics can be plated. Plating can only be performed on specific grades of certain plastics, and the number of these is quite limited. If plating is required, the material suppliers should be contacted to determine which materials can be used. Also, information should be obtained regarding any special requirements or limitations present for the final selected material. It may be possible to consider vacuum-metallizing instead of plating. It is usually less expensive and can be performed on a greater variety of plastics.

Painting is the most common method of decorating a plastic surface (other than molding-in the color). And painting, like other applied finishes, requires proper surface treatment for successful application. Some plastics, such as the polyolefins, may require such drastic surface treatment as *flame treating* to oxidize the surface and allow the paint to adhere. Other plastics, like some of the ABS grades, can be painted directly after being ejected from the mold with little, if any, surface preparation as long as mold releases are not used during the molding process.

Any product design that does not allow proper draft, has deep, thin wall sections (e.g., ribs), or contains other elements that inhibit easy ejection from the mold, creates a condition in which a mold release agent is necessary to aid in ejecting the part. Despite advertising claims, mold releases create a surface on the molded part that is difficult to paint. In most cases, paint will not adhere properly and the finish will peel easily or simply flake away. The designer can greatly minimize the use of mold release by following proper design guides listed in this book, or those available from material suppliers for specific types and grades of materials. And if the product does require any post-molding applied finish, the designer can state on the drawing that no mold releases are allowed during the manufacturing processes of that product.

PACKAGING REQUIREMENTS

Packaging is a manufacturing process that seldom gets attention from the primary product designer. Yet, it can be just as important as any other function the designer must consider. For the molder, the most economical method of packaging, and the one with the least amount of aggravation, is the “bulk” packaging

method. This simply requires that the molded product be dropped into a container (usually a cardboard box or barrel) with no protective covering (not placed in individual protective cells). This method, if acceptable, is the least expensive method of packaging. Actual part content numbers will vary from container to container, and the molder will usually simply weigh the finished container to determine how many parts are being shipped. Of course, the molder can also utilize a simple counter at the machine to record how many parts are being placed in the container, but normally a weigh count is still used to verify that number.

The product designer may have some concerns regarding the bulk method of packaging. For example, the specific product design may contain delicate features such as snap fittings or molded-in inserts that can be easily damaged if mistreated. Or the surface may be decorative and have aesthetic functions which would be jeopardized by falling into a box. In any case, the designer can elect to mandate the method of packaging by stating that each part must be protected during packaging. This can be accomplished by placing a plastic sleeve over the part, or placing each part in an individual “cell” in the container, or a variety of other methods. But if the design does not warrant the extra cost of special packaging, it should not be specified.

Another idea the designer can utilize to minimize packaging costs is that of “nesting” of parts, one within another. This allows many more parts to fit in a carton and keeps the packaging costs to a minimum.

A discussion should be held with the molder (before requesting a quotation) to negotiate proper packaging methods. Protective devices should be discussed, if needed, and the designer should express any concerns about the basic product design. Once the molder is aware that there are some concerns, he or she will usually strive to include those concerns in any future thoughts or discussions regarding packaging methods.

TESTING REQUIREMENTS

It is important to recognize the need for testing as it relates to specific product designs. The product designer must include in his or her list of “things to do” a study of the tests that might be required by law for a specific design. For example, any material used in a product that will be used in an office environment must be able to withstand a specific test for flammability as determined by Underwriter’s Laboratory (UL). If that same product is intended for use in Canada, the material is subject to testing to an additional set of standards as determined by the Canadian Standards Authority (CSA). The CSA standards may be more difficult to pass than the UL standards, or vice-versa, depending on the test used and the expected results. The material supplier is usually the one to submit a

material for UL approval, and once that approval is obtained, the UL releases a form called the “yellow card” that designates the flammability rating based on a specific thickness of a test sample. This card is on file with the material supplier and copies are readily available for a customer’s use. The product designer should make an effort to use materials that are yellow-carded for any product requiring specific UL approval. UL designates such approval by allowing the designer to place the UL seal of approval on a visible portion of the product (as a stamp, or molded-in symbol), or on a tag attached to the product that cannot easily be removed.

Even though the designer is not expected to provide actual test apparatus, it can be beneficial for the designer to note, on the product drawing, what tests will be required before the product is deemed suitable for use. If every part must be tested (although this is very unusual) this also should be noted. Then the molder is aware of the critical nature of the part and will take extra steps to ensure that processing parameters do not jeopardize the qualifications for testing. This will cause the cost of the finished product to increase, so it should not be requested unless absolutely necessary. In most cases, only random testing is required. Once a molding process has been established, it can be maintained to a degree that ensures minimum potential for change in the finished product, if proper equipment and tooling are utilized, and the molder has the required expertise for a specific material.

For regulatory agency approvals, various testing methods must be employed. Each agency has its own requirements and should be contacted directly for information regarding those requirements. The Society of Plastics Engineers (SPE) can be contacted for information regarding the location, addresses, and other contact information for these various agencies.*

SUMMARY

Certain considerations affect the requirements for a specific product design. These include the determination of annual volumes, material handling requirements, secondary operations, packaging requirements, and testing requirements.

The main factor needed to determine the number of cavities required for a specific product is the annual volume requirement. It is necessary to be as accurate as possible with this estimate because molds are expensive, and the more cavities required, the more expensive the mold.

It is prudent to bear in mind that, for a normal 5-year life span, product volume requirements ramp up from just a few in the most early stages of production, to approximately 50% of estimated maximum requirements by the end

*SPE, 14 Fairfield Drive, P.O. Box 0403, Brookfield, Connecticut 68040-0403.

of the first year. This will increase to 100% of estimated maximum by the end of the second year. The 100% trend will continue through the third year, but by the fourth year the requirements will drop to approximately 50%, and by the fifth year they will have reduced to 10% or less.

For secondary operations, handling of the molded product may be necessary for such activities as fixturing, locating, turning, machining, and a variety of others. It is possible that some, or all, of these may be performed by automated equipment or robots. In those cases, especially, it is better to design the product to be easily manipulated.

The success of decorating a surface finish of any plastic product is dependent on the plastic itself. Some plastics (especially polyolefins) are difficult to decorate and usually require some type of surface preparation to obtain adequate adhesion of the decoration. There is a big difference between pigments and dyes, and some plastics can be decorated with only one type, while others may not be able to utilize either.

Any product design that does not allow proper draft, or has deep, thin wall sections (e.g., ribs) or contains other elements that inhibit easy ejection from the mold creates a potential need for a mold release agent to aid in ejecting the part. Despite advertising claims, mold releases create a surface on the molded part that inhibits paint adhesion.

Packaging is an area that seldom gets the attention it should from the primary product designer. Yet it can be just as important as any other function the designer must consider. For the molder, the most economical method of packaging, and the one with the least amount of aggravation, is the “bulk” packaging method.

The product designer must include in his or her list of “things to do” a study of what tests might be required by law for a specific design.

QUESTIONS

1. What determines how many cavities will be needed in a mold to produce a specific product?
2. How does awareness of material handling procedures help the designer?
3. What design device focuses ultrasonic energy to the joint area for ultrasonic assembly?
4. What is one factor that determines whether or not a material is suited for ultrasonic welding?
5. Other than molded-in color, what is the most common method of decorating the surface of a molded part?
6. What special surface treatment may be required to allow paint to adhere to polyolefins?

7. How does a product design that inhibits proper ejection from the mold affect the adhesion of an applied finish to the product?
8. What can the designer do to eliminate the use of mold release on a specific product?
9. From the molder's viewpoint, what is the most economical method of packaging plastic parts?
10. What are two methods that can be used to protect a plastic part during shipping?
11. What is one idea a designer can utilize to minimize packaging costs?
12. Name two governmental agencies associated with flammability testing.

Selecting a Vendor

OVERVIEW

At some point during the development process a decision must be made as to where to place the job to be manufactured, either as a prototype or a production unit. Studies should be performed to determine the capability of candidate vendors (even if the job will be manufactured in an in-house facility) to perform the activities needed to produce a product that will meet time, cost, and quality requirements established during the product development process. Not all vendors are equal in their abilities to provide all things for all customers. For instance, some injection molders are very capable of producing household products such as measuring cups and plasticware from commodity resins, but have neither the equipment nor the expertise to produce tight tolerance, tiny electronic modules using engineering materials. Likewise, the molder that specializes in tight tolerance engineering products cannot be competitive when it comes to producing housewares. It is not fair to either molder to request quotes from them for both types of products. The resultant quotes will not properly reflect their strong and weak points, and there is a definite risk of obtaining inaccurate information concerning the expected quality, cost, and turnaround time for manufacturing the final product. What is needed is an understanding of what is actually required in the molded product, and then finding a suitable vendor to produce it. This need not involve an undue amount of work, and there are certain items to consider that will make the task somewhat easier and much more efficient.

LIST OF REQUIREMENTS

The first step is to make a list of items required from the vendor. Following are some suggestions with appropriate comments.

Dedicated Specialty or Full Service?

Dedicated Specialty

Most vendors today do *not* provide full service, but instead specialize in a particular type of product, material, or process. That is fine as far as it goes, but most molded products require a variety of processes before the finished product

is ready for market. For instance, the part must first be molded, so a mold must be built. And the mold must be designed before it is built. In many cases the mold is designed by one company, built by another, and then run by a third. And, if secondary operations such as painting are necessary, quite often the part must be sent to a fourth and possibly fifth company.

Of course, every time an activity takes place or an operation is performed, there is chance for an error, and it is possible that a mistake occurs which will result in producing defective parts if it is not corrected. The quandary that occurs for the customer at that point is: Who is responsible for the mistake and resultant defects? Often, there is a confusion of finger-pointing and accusation which results in everyone being upset with each other, with the customer absorbing the loss because no one takes responsibility.

Full Service

In the case of a “full-service” vendor, the customer deals with only one vendor. This is usually the vendor who will be responsible for providing the finished product to the customer, usually the molder. The customer explains to the molder what is needed and the molder makes all the arrangements to get it done. If the molder has all of those capabilities in house, fine; there is never any question of responsibility between the molder and the customer. But, if the molder must go outside for certain activities or processes, the molder *still* takes full responsibility for success or failure of the finished product. That is why it is referred to as “full service.” It is intended to help the customer get what is needed without the hassle that sometimes goes with obtaining a finished product. The full-service vendor takes care of any problem and works out the details of responsibility, without the customer getting involved.

It is certainly possible that a customer does not need, or desire, a full-service vendor. This may be the case, for instance, if the customer has a good in-house molding facility, but does not possess a mold design or build operation. In such a case the customer would probably have a mold designed and built externally and then brought in house to actually perform the molding operation. This presents an entirely new set of problems when it comes to accepting responsibility. The moldmaker depends on the mold designer to create a design that will meet the customer’s requirements. This includes items such as determination of shrink factors (in all directions and in all areas), proper selection of mold steel type, authorized gate and ejector-pin locations, and a host of other items. The moldmaker makes the assumption that the mold designer has obtained approval from the customer for these things, if necessary. On the other hand, the mold designer assumes that the customer has selected a moldmaker who understands the customer’s needs and is capable of building a mold from the normal information provided by the mold designer. There is potential here for a limited amount of communication to take place between the two vendors because they

both make reasonable assumptions about the other. When the finished mold arrives at the customer's facility for tryout, the molded parts may not meet the customer's requirements. An immediate battle will occur, with the customer blaming the moldmaker and the moldmaker blaming the mold designer. The mold designer then blames the moldmaker and the customer. And it is possible that to resolve the problem, the only recourse the customer has is to pay more money and take up more time to have "someone" fix the mistake. In fact, this may take up so much time and money that the project fails due to missing the original window of opportunity.

To solve the problem of responsibility when working in such a situation, it is wise for the customer to take as much responsibility as possible for the entire project. While this may seem unwise at first, the result will be that the customer gets what is needed, and the total cost will usually be quite a bit lower (by 20% or more). The reason for this is that the vendors will be able to quote lower prices because they do not have to worry about factoring their costs to include rework. The customer agrees, in advance, to pay if rework is needed. However, the customer must be capable of providing the right information to the vendors, such as shrinkage factors, mold steel types, and other data, as mentioned earlier. If that is not possible, the customer is better off simply working as closely as possible with all the vendors involved, holding frequent joint meetings and making numerous tracking visits to ensure that all vendors are communicating and working at the same level of cooperation. Even then, errors will occur. The customer and vendors should agree beforehand who is responsible for what phase of the project. This approach will minimize the hassle and bickering that will surely occur before the project comes to final fruition.

Usually, it is advantageous both economically and timewise, to select a full-service vendor over a group of dedicated service vendors. However, a variety of factors must be considered before making this kind of determination. Financial concerns will probably prevail, but the factor of responsibility must not be overlooked. If a vendor is not willing to accept responsibility for what is normally considered his area of responsibility, it should be viewed as a red flag that leads to finding another vendor who will.

WHAT TO LOOK FOR

If outside vendors supply molded plastic products to your company, you need to have an organized way to select and qualify such vendors.

While the following list applies to molders, it can also apply to other vendors. It is a checklist designed to minimize your efforts while maximizing the results. The list consists of concerns in three areas—basic investigation, familiarity, and capabilities—and is intended to provide guidelines rather than be all-inclusive.

Basic Investigation

Note Condition of Plant and Equipment

During your initial survey trip (and be sure to make a personal visit), pay close attention to the overall cleanliness of the plant and condition of the equipment. A messy plant, or equipment with rags tied around oil leaks, indicates lack of concern for details. This can result in late deliveries and poor-quality products.

Ask to See Product Samples

Reputable vendors are proud to display the types of products they have produced. Don't expect to see the failures, but observe the quality level and complexity of the products offered. Usually, these will be representative of what you can expect for your own product.

Check Delivery Performance

Ask to see specific examples and records of deliveries made versus promised dates. While most promised delivery dates *do* change, the changes can usually be explained by alterations in customer requirements, formal engineering changes, or natural disasters, none of which are the fault of the vendor. Beware of any vendor who has no good explanation for missed deliveries. This indicates a potentially dangerous trend.

Check Quality Rating

Ask to see awards, letters, or documents stating the quality rating of the vendor by other companies, or other divisions within the same company. Most vendors are proud to display these items and, while absence of such does not necessarily demonstrate poor quality, it can indicate that quality may not be of utmost importance to that vendor.

Investigate Specialty Areas

Every vendor has an area of expertise in which it excels. This may be large parts, quick turnaround, low cost, high quality, a variety of other capabilities, or a combination of these. No single vendor is capable of producing all types of products in all types of materials, using all types of processes, to all levels of quality. In any case, there should be a match with the specific product requirements at hand. This is especially true when considering full-service vendors.

Determine Equipment Availability

Don't be overly concerned with the age of equipment. Older machinery is quite capable of producing quality products. But do be concerned with the availability of the equipment. Most vendors experience periods of bustling activity alternated with a series of lulls. This is common. It is important to understand how those cycles may affect specific products that are being considered for placement with specific vendors. Also, find out if overtime is available, and at what cost.

Examine Bidding Practices

We all want the biggest bang for our buck. But be careful. Low cost does not mean inexpensive. How does the particular vendor usually fare in the bidding process? A vendor consistently bidding outside the high or low range of other comparable vendors may not be in a position to properly manufacture those products. This may be due to such factors as an overcapacity at the time of quoting, a misunderstanding of what the customer requires, or an inability to estimate charges competitively. Make sure you are requesting quotes from vendors capable of producing your type of product.

Familiarity

Normal Tolerance Levels

All vendors work well within specific dimensional tolerance requirements. Some prefer very tight tolerances, while others prefer loose, or standard tolerances. The product design will dictate the tolerance level required, and a vendor should be chosen that works comfortably within that level. Of course, because tighter tolerances cost more, the product designer should always specify the loosest level of tolerance acceptable for a specific product design.

Materials

Although some vendors advertise that they are capable of manufacturing products made from any plastic, most are willing to admit their limitations. Remember, there are over 20,000 materials available to the product designer today. There are many variations between them in the areas of molding parameters and dimensional control. No vendor can be expected to be able to mold all of them. A vendor should be willing to show an established expertise with the specific material required by the product design. This can be verified through various samples, or letters of commendation from material suppliers, or customers.

Molding Processes

Some vendors specialize in only one type of process, while others have the equipment for multiple processes. It is important that the vendor be able to exhibit specific expertise in the process required for any specific product being considered for placement. For example, a vendor with expertise in standard injection molding may not be capable of satisfactory insert molding.

Equipment Type, Age, and Size

Although older equipment *can* be quite capable of producing many different product designs, state-of-the-art equipment sized for a specific product design and capable of producing the specific product at the lowest possible cost with the highest level of quality, is critical to a successful program. Compromising any of these items means compromising the cost or quality of the product.

Subcontracting

Full-service vendors will provide operations they do not perform themselves by subcontracting to another vendor. Responsibility for the quality, cost, tooling, and delivery of that contracted service should be established as belonging to the prime vendor, although the customer may wish to certify the subcontractor to achieve a high level of overall confidence in the success of the total project.

Capabilities

Volumes

A vendor must prove the capability of producing the required number of products in the required length of time, as specified by the customer. A schedule must take into consideration production time as well as transportation and tool delivery times. The customer should discuss with the vendor the plan for how and when products will be delivered. Also to be considered are the cyclic tendencies of the existing work that the vendor is molding. It may be that the vendor is extremely busy at certain times of the year and these times may conflict with your specific requirements, thus potentially jeopardizing your schedules.

Tool Design, Build, and Repair

Some vendors design and build the tools required for a specific product, while others subcontract some or all of these activities to others. Most primary vendors have repair facilities on site or close by, but these issues must be considered when an initial vendor is selected. If work is to be subcontracted, a thorough understanding of final responsibilities must be established.

Computerized Operations

A vendor that has elected to use computer-assisted operations wherever possible can be considered one that wishes to ensure accurate processing of materials, toolmaking, traceability, and office procedures, including scheduling and quality control. Electronic data transfer (EDT) capabilities between the vendor and the customer are valuable tools for ensuring accurate, fast transfer of information and quick response time. In most cases, computer-generated drawings can be downloaded directly from the customer to the vendor and then directly to toolmaking or processing equipment, thus reducing overall product development cycles.

Statistical Process Control (SPC) and Other Quality Programs

If a vendor is not utilizing SPC today, it can be an indicator that the vendor is reluctant, or unable, to provide the customer with the expected product quality level at a reasonable cost. Other programs and systems can be used to provide the information, but the important thing to consider is how well the vendor

is able to track, document, and implement performance efficiency and quality levels. Another item to look for is whether or not the vendor has considered ISO 9000 registration. If not, it can mean they are not financially secure enough to afford the full program, or they are not interested in performing the work and allocating the resources to obtain registration. Either case may indicate a reluctance to provide the necessary activities to ensure the highest quality parts at the lowest possible cost to its customers. If a vendor has already received ISO recognition, check to see how long it has been in effect. If more than a year, ask to see their recertification documentation. If there is none, it probably means they have not met the evaluation requirements to continue in the program.

Responsibilities

Listed below are seven primary areas of responsibility that should be considered mandatory for discussion and negotiation between a customer and a vendor. While these do not reflect *all* the possible responsibility areas, they do cover those of primary concern. It is good practice to include the final, negotiated terms regarding these responsibilities in the purchase order package.

1. Suggestions to the customer for changes in initial product design, or material selection, to ensure a quality product that is readily manufacturable. This would include items such as elimination of sharp corners, gradual versus abrupt wall thickness changes, adequate draft allowance and tolerances, etc.
2. Early assistance with primary and secondary tooling “design-and-build” programs to ensure that all tooling will produce quality parts for the expected life span of the product.
3. Acceptance of responsibility for maintaining all associated tooling and equipment in good repair, according to professional standards, for the life of the contract.
4. Establishment of an in-house material inventory control system to ensure the use of proper raw material in the manufacturing of the customer’s product.
5. Use, maintain, and periodically calibrate all necessary process controls, fixtures, and inspection equipment needed to ensure a quality product.
6. Maintain proper inspection procedures and records to allow full traceability of failures, rejects, defects, and acceptable products.
7. Establish and maintain an incoming raw material inspection, testing, and quality control program.

In addition, the success of any manufacturing program depends to a great extent on the willingness of the vendor to participate and cooperate with the customer in establishing specific parameters, and controlling, reporting, and auditing the methods and records pertaining to the customer’s products.

QUALIFICATION FORM

Figure 14-1 is a specially-designed form that can be used for qualifying a potential plastic molding vendor. It is being used with the permission of Texas Plastic Technologies,* designer of the form. They give permission to the reader to copy the form and use it for noncommercial purposes.

Date _____		
Vendor name _____		
Division of _____		
Street address _____		
Address two _____		
City _____	State _____	Zip code _____ - _____
Phone _____	FAX _____	
E-mail address _____	URL _____	
Executive and Key Personnel.		
President _____		
Vice-president _____		
Engineering manager _____	Phone _____	
Production manager _____	Phone _____	
Quality manager _____	Phone _____	
Total personnel _____		
Quality _____	Engineering _____	Production _____ Toolroom _____
A. Molding process capability.		
1. Thermoset (please check all that apply).		
<input type="checkbox"/> Compression molding	<input type="checkbox"/> Transfer molding	<input type="checkbox"/> Transfer/injection
<input type="checkbox"/> Injection	<input type="checkbox"/> RIM	<input type="checkbox"/> Other _____
<input type="checkbox"/> Thermoset specialty _____		
2. Thermoplastics (please check all that apply).		
<input type="checkbox"/> Injection	<input type="checkbox"/> Blow molding	<input type="checkbox"/> Sheet manufacturing
<input type="checkbox"/> Thermoforming	<input type="checkbox"/> Structural foam	<input type="checkbox"/> Two-shot injection
<input type="checkbox"/> Insert molding	<input type="checkbox"/> Overmolding	<input type="checkbox"/> Other _____
<input type="checkbox"/> Thermoplastic specialty _____		
3. Materials most commonly used.		Approximate annual pounds.
a. _____	_____	
b. _____	_____	
c. _____	_____	
d. _____	_____	
e. _____	_____	
f. _____	_____	
<input type="checkbox"/> Check here if you have mechanized color mix capability		

Figure 14-1. Plastic molding vendor qualification form.

*Texas Plastic Technologies, 605 Ridgewood Rd., Georgetown, Texas 78628.

4. Molding equipment.

Brand	Year built	Tons	Ounces	Number
_____	_____	_____	_____	_____
_____	_____	_____	_____	_____
_____	_____	_____	_____	_____
_____	_____	_____	_____	_____
_____	_____	_____	_____	_____
_____	_____	_____	_____	_____
_____	_____	_____	_____	_____

Comments _____

5. Type and method of drying equipment.

6. What type of process controllers do you have on your presses?

B. Finishing/secondary process operations (please check all that apply).

Process	Manual	Automatic
<input type="checkbox"/> Painting	<input type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/> Silk screening	<input type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/> Hot stamping	<input type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/> Pad printing	<input type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/> Sonic welding/inserting	<input type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/> Plating	<input type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/> Vacuum metallizing	<input type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/> Drill/tap/machine	<input type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/> Assembly operations	<input type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/> Other _____	<input type="checkbox"/>	<input type="checkbox"/>
_____	<input type="checkbox"/>	<input type="checkbox"/>

Specialty _____

C. Manufacturing specialty (please check all that apply).

<input type="checkbox"/> Miniature functional parts	<input type="checkbox"/> Large functional parts
<input type="checkbox"/> Small decorative parts	<input type="checkbox"/> Large decorative parts
<input type="checkbox"/> Electronic parts	<input type="checkbox"/> Precision molding
<input type="checkbox"/> Insert molding	<input type="checkbox"/> Two-shot molding
<input type="checkbox"/> Prototypes	<input type="checkbox"/> Engineering materials
<input type="checkbox"/> Other _____	
<input type="checkbox"/> Preferred _____	

D. Production volumes.

1. Preferred annual volume range _____
2. Minimum annual production volume _____

Figure 14-1. (Continued)

E. Company specialty (tell us about your specialties).

F. Mold manufacture and repair.

1. How many moldmakers do you employ in your shop? Repair _____ Build _____
 2. What level of repair can you perform?
 - a. Minor (replace pins, add vents, etc.). ☐ Yes ☐ No
 - b. Medium (alter gates, rework waterlines, etc.). ☐ Yes ☐ No
 - c. Major (alter cavities/cores, repair parting line, etc.). ☐ Yes ☐ No
 - ☐ We don't do repair at this facility.
 3. Do you build molds at this facility? ☐ Yes ☐ No
 4. Do you have mold design capabilities? ☐ Yes ☐ No
- If yes, which programs? _____

G. Record keeping

1. What type of molding records are kept? _____
2. What department is responsible for keeping these records updated? _____
3. How long are records kept before archiving or destroying? _____
4. Do you maintain machine setup records? ☐ Yes ☐ No
5. If yes, what department is responsible for keeping these records updated? _____
6. Can you trace an individual part back to the lot of material from which it was molded? ☐ Yes ☐ No
- If yes, what method is used? _____
7. How do you track rejected parts and defects? _____
8. How do you track rejected material? _____
9. Do you keep samples of each lot of material? ☐ Yes ☐ No
10. If yes, how much and where is it kept? _____
11. What method of raw material storage do you employ? _____
12. How do you track material from storage to the finished part and what type of labeling system, if any, do you use? _____

Figure 14-1. (Continued)

H. Inspection.

1. What type of test and inspection equipment do you have for molded parts?

2. What type of test equipment do you utilize for raw material?

3. If you employ a “zero-defect” program, please define it and describe how you apply it to molded parts.

I. Certification.

1. How do you certify that a “lot” of parts meets all the material and dimensional specifications?

2. How do you certify compliance of your incoming raw material?

3. What is your mold approval procedure for new molds?

4. What is your mold approval procedure for reworked molds?

5. What is your policy and approval procedure for molds transferred to you from another source?

6. Have you obtained ISO 9000 registration?

☐ Yes ☐ No

7. If no to #6, is it, or a similar program, being pursued?

☐ Yes ☐ No

8. If yes to #7, when do you anticipate completion? _____

J. General comments.

Prepared by _____ Date _____

Title _____

Figure 14-1. (Continued)

SUMMARY

At some point in the development process a decision must be made as to what vendor (either in house or outside) will be selected to perform the manufacturing activities.

Not all molding vendors are capable of performing all molding processes. Neither are all molding vendors capable of molding all types of materials or product designs.

A selection process will help determine those vendors capable of manufacturing a specific product. First in this process is to determine whether a dedicated-service vendor or a full-service vendor is required, or desired.

There are three areas to consider when performing a survey of a vendor: basic investigation, familiarity, and capabilities.

It is advisable to request representative samples of the type of work a vendor can do. The vendor will not show any failures, but will focus on those products he or she is most proud of and a determination can be made from those samples as to whether or not the vendor molds parts similar to those of the potential customer.

Over 20,000 plastic materials are available. No single vendor is able to mold all of them. All vendors have a group of materials they feel most comfortable with and another group they simply do not wish to mold. Do not hesitate to ask for information on each list.

While new equipment may be preferred, older equipment is certainly capable of producing parts to equal quality standards, especially if the equipment has been regularly maintained or recently refurbished.

In addition, the success of any manufacturing program depends to a great degree on the willingness of the vendor to participate and cooperate with the customer in establishing specific parameters, and controlling, reporting, and auditing the methods and records pertaining to the customer's products.

QUESTIONS

1. What is the quandary the customer faces when he or she selects a dedicated-service vendor?
2. Give an example of when a customer may not wish to find a full-service vendor.
3. For what does the moldmaker depend on the mold designer?
4. List the three areas of concern to consider when selecting a vendor.
5. What does a messy plant indicate, and what is the probable result?
6. Why should you ask to see a vendor's product samples?

7. Why should you not be overly concerned with the age of a vendor's equipment?
8. What does it indicate if a vendor elects to use computer-assisted operations whenever possible?
9. Why is early vendor assistance with primary and secondary tooling design-and-build programs preferred?
10. When using a vendor, on what does the success of any manufacturing program depend?

Prototype Overview

THE VALUE OF PROTOTYPING

After a product has undergone the initial design phases and has been optimized through team involvement, it is time for a decision as to whether or not a prototype should be built. In some cases prototypes can reduce the overall development cycle time, while in other cases the use of prototypes may add to the overall cost and time of the total development cycle, so this course of action, if accepted, must be justified.

Often, a prototype may be required for marketing purposes to show the basic concept or appearance of the final product. This may be the case when a prototype is used for display purposes at an industry exposition. Or a prototype may be used for initial testing purposes to prove the basic design concept and to ensure that the product is capable of withstanding end-user exposure or shipping and handling functions. But usually the primary reason for using a prototype is to achieve a comfort factor for form, fit, function, and appearance of a final product. Following are some of the advantages of using prototypes for these purposes; of course, the disadvantages are cost and time.

Form

The basic shape of a product is certainly important to the designer as well as the end user. Most products are shaped the way they are for a specific reason, such as the shape of a soft drink container that is intended to feel comfortable in the user's hand. Another example is the shape of nested boxes for storage purposes; they must fit together properly to take full advantage of their ability to nest. Building a prototype model prior to investing time and money in final production tooling, equipment, and materials allows the designer to see firsthand if the product as designed will meet the desired intentions.

Although the product can be seen on paper (or computer screen) and can be manipulated there to check the basic shape and form, nothing builds the confidence level like a part that can be touched and held. Here is where a prototype has its greatest value.

Fit

Most plastic products are designed with a specific *fit* in mind. Two or more components may be designed to fit together to make the final product, or the final product may be designed to fit within another product, or another product within the plastic product being designed. In these cases, while the drawing can be optimized for apparent proper fit, it doesn't compare to an actual part that can be tested for all the intended fit conditions. Usually, when a prototype is built for this reason, the product dimensions end up being altered either in their original nominal numbers or in the tolerances applied. This is because it is difficult for most designers (or others for that matter) to imagine tolerances in the range of 0.005 to 0.010 in. (0.127 to 0.254 mm). On the drawing, they may seem enormous to the designer, who wishes no tolerances were necessary at all, and therefore may make them excessively tight. But when the final parts are made, there is awareness of what actual tolerances produce, and, normally, concessions can be awarded. This is one of the values of building prototypes.

Function

All products are designed with some function in mind. A milk jug is designed to hold liquid. A gear is intended to transfer energy. Safety helmets are meant to protect the head of the user. How can one be certain that a specific product design performs the function it was intended to without producing parts? One way is to build a prototype. This prototype does not have to be a hard version of the product, but can actually be a virtual product existing only in a computer file. All kinds of functional analyses can be performed on a virtual prototype. Finite element analysis (FEA) programs for computers are in abundance for determining stress levels, analyzing motion, checking compressive and tensile factors, and just about any other functional attribute imaginable. And if this is not enough, hard copy prototypes *can* be created to check these items further. This situation is certainly preferred to waiting until large investments of time and money are made to manufacture the first production products, only to find they do not function as intended.

Appearance

While a specific color, texture, and contour can be noted and drawn on paper (or screen) for a particular product design, an actual part in hand is the only way to tell for sure what the final product will really look like. There are cases in which proper appearance can make the difference between a successfully marketed product and a failure. A prototype allows the designer to see if the product meets the requirements of appearance before, again, large investments of money and time are made in the production phase. Alterations can be made if necessary

to create a final product that does meet requirements, and building of the final tooling can progress on schedule.

A prototype offers a major advantage when texture is to be used on the surface of the molded part. This is because the selection of a specific texture is usually accomplished by viewing texture samples provided by the company performing the texturing process on the injection mold. These samples cover a range of thousands of possible styles, from a faint satin finish to a heavy alligator hide. However, they are usually provided on a flat black plastic (usually a styrene) that is rarely the same plastic being specified for the final molded product. That material may cause the texture to appear quite different from the sample that was initially viewed, especially if the color is not flat black. A prototype can supply parts that will have that final appearance, and a judgment can be made to keep the texture or change to a different one.

The Prototype Option

Though the decision to build a physical prototype is driven by schedule and budget considerations, prototypes can be made to closely resemble the final product so that form, fit, function, and appearance can be assessed. They also can be made simply for aesthetic reasons so that a product can be displayed for marketing purposes. Resultant changes can be made and the product reevaluated before volume production parts are sourced, tooled, and produced.

TYPICAL METHODS AND RECOMMENDED QUANTITIES

Many methods are available for creating prototype parts, some simple, some complex. We address the more common methods here.

Fabrication

In this process, plastic sheet and rod stock are machined and bonded together to create a facsimile of the desired product. This is one of the oldest methods of producing prototypes and is ideal when time is not critical and cost must be held to a minimum. It is usually cost-effective for up to five or six units. It is not advisable to use these prototypes for structural, thermal, or electrical testing however, because they will react quite differently from those that are injection-molded.

Thermoforming

In thermoforming, plastic sheets are vacuum (or pressure) formed over models of wood, plaster, latex rubber, aluminum, or other easily-machined material, and then cut to final shape and bonded to form the finished product. This method requires very little tool investment, but the models are only good for approximately

two to three dozen units. The cost of additional machining and bonding can significantly add to the final cost. These prototypes can be used for basic appearance and function evaluations, but dimensional accuracy is poor.

Casting

In the casting process, a plastic such as a liquid urethane is poured into an epoxy, aluminum, or similar material mold that was machined to the required product dimensions. The two-part resin system hardens and is then removed from the mold and finish-machined to specifications. This is a good system to consider for up to 50 units from each mold. Because they are cast and finish-machined, dimensional accuracy can be quite good, and structural, electrical, and thermal testing results will closely approximate those of injection-molded parts.

Molding

In this process a plastic resin is injection-molded into a “soft” injection mold made of aluminum, epoxy, polyester, or soft steel to form the finished product. The process approximates the actual production method of manufacturing more than any other prototype method and can be considered when required quantities are in the range of approximately 500 to 5,000 units. If properly molded, these parts can be used for testing and display and even early production while waiting for final production tooling.

Virtual Prototyping

Computer programs are available that allow the product designer to “see” on a computer screen what could only have been seen before with a physical product. The programs utilize finite element analysis to assess conditions created by the initial product design and graphically detail the results so the designer can determine what alterations may be made to create a better final product. This method of prototyping can be used to answer such questions as: What if a different plastic is selected? or, What if support ribs are added and wall thicknesses reduced? or, What if dimensions are changed?, etc. The final design can then be saved in a format that can be downloaded to other programs and equipment used for designing and building molds, or producing hard-copy prototypes, if needed. This method is usually used to produce only a single, virtual prototype.

Rapid Prototyping (RP)

Stereolithography Apparatus (SLA)

SLA is a process in which a laser beam created by an ultraviolet light source is focused on the surface of a pool of liquid plastic called a photopolymer. The energy of the laser causes the liquid plastic to solidify to a thickness of approximately 0.005 in. (0.127 mm) at the point of contact by the beam. Therefore, any configuration that can be drawn by the laser beam can be converted to a solid

object. The process can be compared to the action of an office laser printer and the image it creates on a piece of paper. The difference is that in the SLA machine the 0.005-in.-thick solid object is created on a plate (called an elevator) that then drops 0.005 in. in preparation for accepting the next layer to be created by the laser beam. This new layer fuses to the preceding layer, and the process continues until a complete part is produced. Depending on the size of the part, this process can take from 2 to 48 hours or more. The advantage is that a finished part can be produced in a very short time from initial conception. The SLA unit takes data generated during the design phase and uses that data to create the finished part. The disadvantage is cost. These parts are extremely expensive. Therefore the process should be considered only when it is vital to obtain prototype information in a short period of time. The process can be used for quantities from a single unit to untold multiple units.

Selective Laser Sintering (SLS)

The SLS process is similar to SLA but instead of a liquid pool of plastic, SLS uses a thin layer of fusing powder that is spread across the surface of the elevator plate. A laser beam traces the area to be created and sinters the powder to a hard material as it does so. This layer is then lowered and a new layer created and fused to the first. The process continues until an entire object is completed. This process also uses CAD data generated during the product design phase. Because it is expensive, this process should be considered only when time is critical and prototype information is required in a short period of time. The process can be used for quantities from a single unit to many multiples.

Laminated Object Manufacturing (LOM)

Again, the LOM process is similar to the SLA process, but instead of using a pool of plastic, LOM uses specially-created paper-stock sheet from a continuous roll. This sheet is pulled across an elevator plate and placed in position. A laser beam, using CAD-generated design data, traces a pattern across the top of the sheet and cuts out that pattern from the roll stock. The sheet is “cured” everywhere the beam touches. The roll indexes to a new area, the elevator drops, and another layer is created over the first. This process continues until a complete part is created.

Ballistic Particle Manufacturing (BPM)

The BPM process builds objects by spraying tiny molten particles of a specially-formulated (proprietary) thermoplastic material with a 220 to 250° F (104 to 121° C) temperature range. These particles are deposited with a piezoelectric jet firing at 12,000 cycles per second, onto a platform where the droplets immediately solidify. The five-axis robotic head can build hollow as well as solid objects.

Solid Ground Curing (SGC)

The SGC process involves combining additive and subtractive layers of patterns using an ultraviolet light source to expose a photosensitive (curing) liquid plastic material, similar to that used in the SLA process. After each layer is created, the excess resin is vacuumed and voids that were created in the object are filled with liquid wax which then hardens. This resin/wax combination layer is then milled down to a precise thickness, a new layer of resin is applied, and the process continues until a completed part is manufactured. After the part is complete, the hardened wax is removed using a mild acid solution.

Which is Best?

New methods of creating prototypes are constantly being developed. They should all be investigated for appropriate use concerning specific applications and product designs. Size, cost, accuracy, or time limitations may suggest that one process is superior to another. One should even consider the older, standby processes such as modeling with clay or machining from wood or other material. The correct process is determined by the designer's requirements. A sampling of RP equipment manufacturers is shown in Table XV-1.

Table XV-1. Rapid Prototype Equipment Manufacturers

<p>BPM Technology, Inc. <i>(Ballistic Particle Manufacturing)</i> 1200 Woodruff Rd. #A-19 Greenville, SC 29607 Phone (803) 297-7700 Fax (803) 297-7711</p>	<p>Cubital America, Inc. <i>(Solid Ground Curing)</i> 1307F Allen Dr. Troy, MI 48083 Phone (313) 585-7880 Fax (313) 585-7884</p>
<p>DTM Corp. <i>(Selective Laser Sintering)</i> 1611 Headway Circle Bldg. #2 Austin, TX 78754 Phone (512) 339-2922 Fax (512) 339-0634</p>	<p>Helisys, Inc. <i>(Laminated Object Manufacturing)</i> 24015 Garnier St. Torrance, CA 90505 Phone (310) 891-0600</p>
<p>3D Systems <i>(Stereolithography Apparatus)</i> 26801 Avenue Hall Valencia, CA 91355 Phone (805) 295-5600 Fax (805) 295-0249</p>	

SUMMARY

The principal reason for creating a prototype is to ensure the desired form, fit, function, and appearance of a final product.

Some prototypes are used for testing purposes, and some may be used as marketing tools that can be displayed at shows or customer sites.

Though the advantages of using prototypes are many, they are tempered by the disadvantages of cost and time to build.

The common methods of creating prototypes include fabrication, thermoforming, casting, molding, virtual prototyping, and rapid systems.

New methods of creating prototypes are constantly being developed. They should all be investigated for appropriate use concerning specific applications and product designs. Size, cost, accuracy, or time limitations may suggest one process over another. One should even consider the older, standby processes such as modeling with clay, or machining from wood or other material. The correct process is determined by the designer's requirements.

QUESTIONS

1. What is the primary reason for building a prototype?
2. What are the disadvantages of using prototypes?
3. How can one be certain that a specific product design performs the function it was intended to without producing parts?
4. Give an example of how a plastic product is designed with a specific "fit" in mind.
5. What do the letters FEA stand for concerning prototype programs?
6. What are FEA programs used for concerning prototypes?
7. Why is proper appearance of a plastic part so important?
8. What styles of texture samples are available for selection?
9. In a single sentence, describe the "fabrication" method of creating a prototype part.
10. What do the letters SLS stand for regarding rapid prototyping systems?

A Primer on Molds

DESCRIPTION OF THE MOLD

The injection mold is the heart of the injection molding process. It is where all the forming action takes place. The molten plastic material, which has the consistency of warm honey, is injected into the mold, under pressure, where it takes the shape of whatever the mold looks like inside (the cavity image). The material is then cooled until it becomes solid again. When it is ejected it has the exact image of the shape inside the mold. In this chapter we look at the injection half of the mold, the “A” half. Here we see how the plastic material is injected into the mold and how the finished shape is formed. We then examine the ejector half of the mold, the “B” half, from which the finished product is ejected.

The “A” and “B” Plates

The mold consists of several different components, but the prime components are the “A” and “B” plates, shown in Figure 16-1.

These are the “A” and “B” plates for a two-cavity mold. Note the two triangle-shaped areas in the “B” plate. These are called the *cavity images* because they have the shape of the product the mold is going to produce. In this case that will be two identical plastic triangles with each side 6 in. (152 mm) long, and the parts will be 1/8 in. (3.2 mm) thick. So that is the shape that must be carved (or machined) into the “B” plate, twice. The “A” plate will just be flat. However, in some cases the “A” plate will also have part (or all) of the image machined into it, if the design of the product dictates it.

The Cavity Image

The cavity image is machined into the plates using standard machine shop equipment such as milling machines, grinders, drills, reamers, and other equipment. Often a machine called an *electrical discharge machine* (EDM) is also used. All of this equipment is capable of producing an image to very exact dimensions, and any image that can be photographed is capable of being copied by machining.

Because of the skills required to operate the machining equipment, and the large investment required to buy the equipment in the first place, the machining

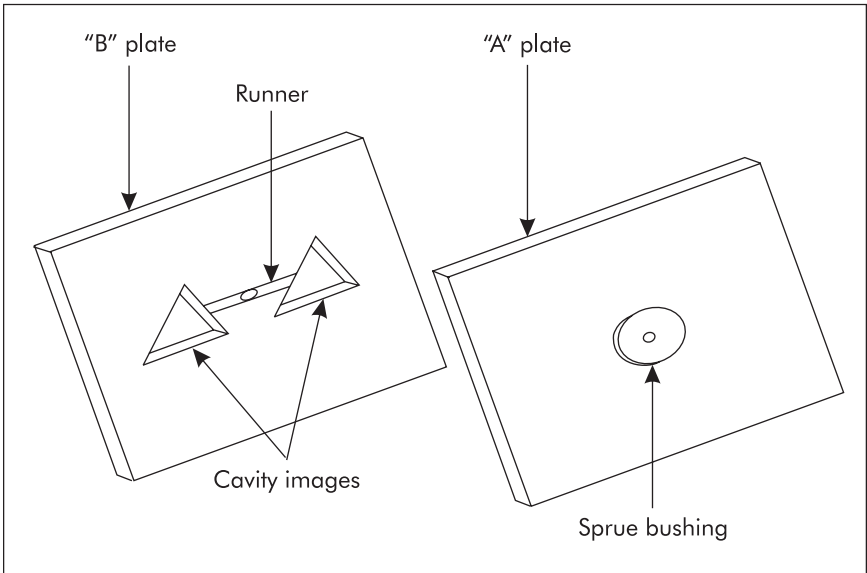


Figure 16-1. "A" and "B" plates of a two-cavity mold.

of molds is an expensive operation. Finished molds can cost anywhere from a few thousand to a few hundred thousand dollars, depending on their size and the complexity of the cavity image. That is why it is so critical that molds be treated properly.

For instance, suppose a molded part does not eject all the way out of the mold and gets stuck. If the mold closes again, that stuck part will probably break some of the ejector pins, core pins, cams, or other fragile components of the mold. That breakage will cost hundreds or thousands of dollars to repair. Moreover, the mold will have to be removed from the molding machine, which will result in lost money to the molding company because parts are not being produced. So it is imperative that the mold be visually inspected every time it opens to make sure the part is properly ejected. At the same time, a quick look will determine if anything is in the mold that should not be there, such as too much flash. If there is a problem of any sort, *the gate should not be closed*, because that will start the cycle over again, and the mold will close on the obstruction.

THE INJECTION HALF OF THE MOLD

The injection half of the mold contains the *sprue bushing* that allows the melted plastic to enter the mold and fill the cavity image, producing the finished molded

plastic part. It is mounted in the center of the locating ring which aligns the mold to the nozzle of the machine's injection unit.

The Sprue Bushing

The job of the sprue bushing is to seal tightly against the nozzle of the injection barrel of the molding machine and allow molten plastic to flow from the barrel into the mold. There is a tapered hole in the center of the sprue bushing through which the plastic flows. The hole is tapered to allow the plastic, after it solidifies, to be removed easily to prepare for the next cycle. There are many different basic designs for sprue bushings, the most common being that shown in Figure 16-2. The next most common does not utilize a radiused seat for meeting the machine nozzle, but rather has a flat face. In the case of vertical molding machines, the sprue bushing is usually split lengthwise and therefore can have a straight internal hole rather than a tapered one.

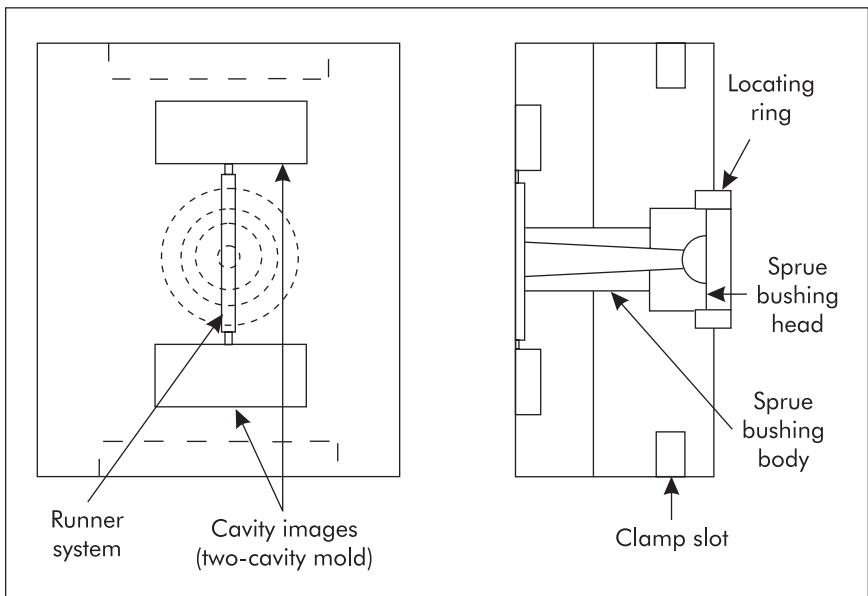


Figure 16-2. The “A” half of the mold.

Runners

In a conventional mold, the sprue bushing directs molten plastic to the cavity images through channels machined into the face of the “A” and “B” plates. These channels allow plastic to run along them, so they are referred to as *runners*.

To save material and cycle time many molds are built with *hot* runner systems, which are designed to keep the plastic material molten right up to the cavity image. The molten plastic runner is captured in steel and stays molten between cycles, thereby reducing the amount of material required to produce the plastic parts during a single cycle.

Flash

The clamp unit of the press is designed to exert enough force against the mold halves to keep them closed during the injection process. Sometimes, if injection values change for some reason, the injection pressure may overcome the established clamp pressure. When that happens, the mold will open a small amount during the injection phase, and a moderate amount of melted plastic will seep out. This seepage is called *flash*. Flash is thin, usually in the range of 0.002 to 0.005 in. (0.050 to 0.127 mm) thick. This is the approximate thickness of a piece of thin writing paper. Although the flash is thin it can be extremely hard and will cause damage to a closing mold if it flakes off or becomes stuck on the mold surfaces. If a mold is flashing it should be corrected immediately. However, a small amount of flash may be allowed as long as it is cleaned off before closing the mold. This should be allowed only in special cases (such as short-term critical scheduling), and never as a normal operating condition.

As a mold becomes old and worn, it may begin to flash slightly. In many cases, this is okay; company managers may have decided that it is more cost-effective to let it flash until they can justify the cost of repairing the mold, as long as no damage is done. In such a case, it may be necessary to remove the flash from the molded product by using a knife-like tool that peels the flash. This is not to be considered a normal situation and the flashing condition must be fixed at some time in the near future or the mold will only get worse and the part quality will steadily deteriorate.

THE EJECTOR HALF OF THE MOLD

Ejector Pins

When the mold opens, the finished part is pushed out of the mold by a number of ejector pins shaped like nails. They have a head, but instead of a sharp point at the other end, they are flat. This flat face is what pushes against the molded part. Figure 16-3 shows how they are constructed.

The pins have three areas. The *face* is the part that pushes directly on the

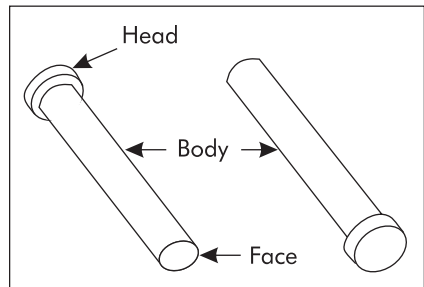


Figure 16-3. Ejector pins such as these facilitate part ejection.

plastic product. The *body* is the stem portion of the pin, and the *head* is the part that keeps the pin locked in the mold itself.

Ejector Plates

There are two plates within the mold that lock the heads of the pins and keep them from coming out of the mold (Figure 16-4). The plates are identified as the *ejector bar* plate and the *ejector retainer* plate. The retainer plate holds the heads and the bar plate is bolted against the retainer plate to keep the heads in place.

Figure 16-5 shows the ejector plates mounted in the “B” half of a mold base.

Knockout Rod

Note the *knockout rod* in Figure 16-5. This is attached to the molding machine and enters the mold base through a hole or holes in the ejector housing. There,

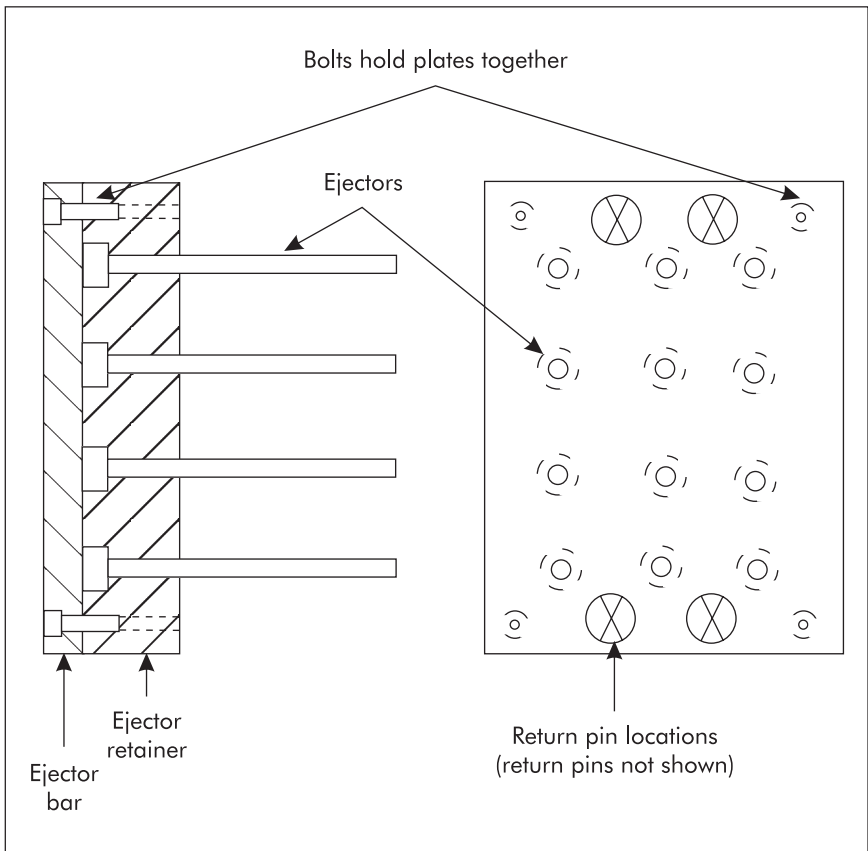


Figure 16-4. Ejector plate device showing ejector pin locking concept.

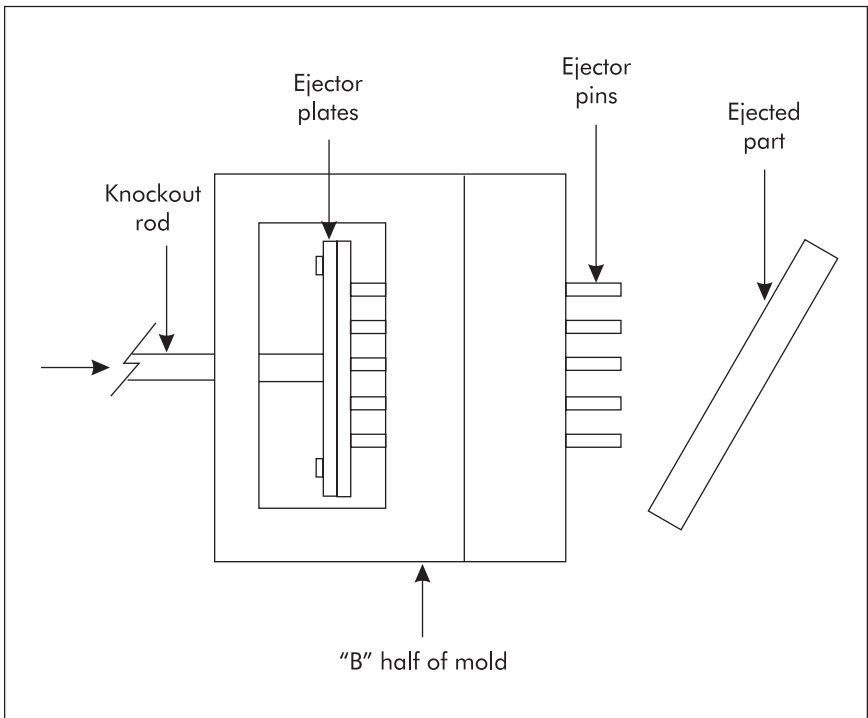


Figure 16-5. Ejector plates actuated by knockout rod.

the rod pushes against the ejector bar plate to advance the ejector pins which push the finished plastic out of the mold. The knockout rod is also called the *ejector rod*, or *ejector bar*. This design is typical but there are mechanisms other than the knockout rod that may be used to perform the same function, such as chains, outboard rods, and hydraulic cylinders.

MOLD DESIGN BASICS

Although mold design is covered in detail in Volume I of this series, it is important to address a few items of major concern. These have to do with gate location, runner design, and venting concepts.

Gate Location

To minimize stress, it is better to locate a gate so that the molten plastic enters the cavity image at the thickest section of the part (Figure 16-6). Although it is ideal to have parts with even wall thicknesses throughout (in which case the gate can be located anywhere), most parts have thin and thick sections.

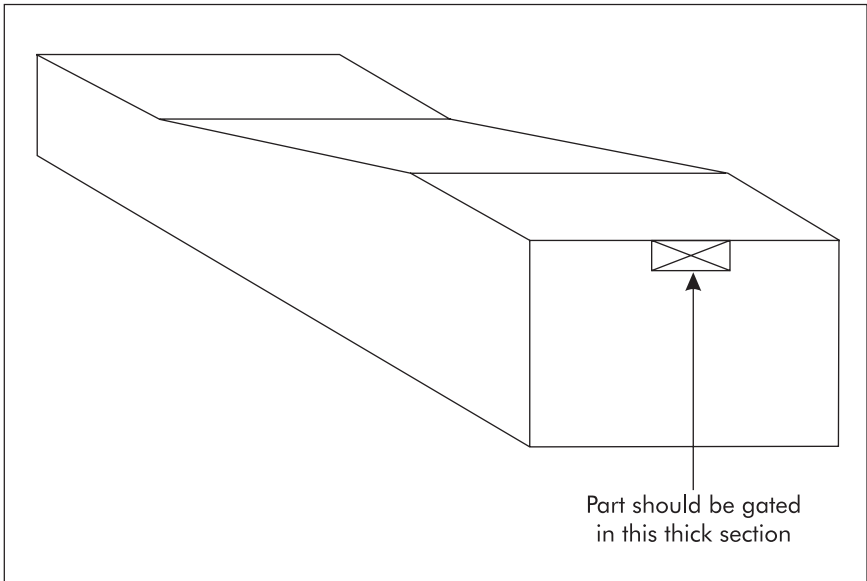


Figure 16-6. Locating the gate at the thickest section of the part reduces potential for stress.

As molecules flow from a thin section to a thick section, they are first compressed and then allowed to expand to take up the space available to them. The gate should be located at the thick section to allow the molecules to gently compress (but not expand, because the fluctuating expansion of molecular size is one of the major causes of stress).

Runner Cross-section

A full round runner is ideal. This is because a circular cross-section creates equal pressure in all directions on the plastic molecules, while a noncircular section causes unequal pressure. This is demonstrated in Figure 16-7 which compares a full round runner design to a common trapezoid design.

Using the runner design on the left will minimize the amount of molecular distortion created while the molten plastic is flowing through the runner toward the cavity. This molecular distortion results in stresses in the material; stressed molecules are carried into the cavity, where they solidify in their stressed state.

Venting Concepts

A large amount of air always becomes trapped in a mold when the mold is closed in preparation for the injection phase of the molding process (Figure 16-8). This air must be displaced, or removed, so that incoming plastic material will

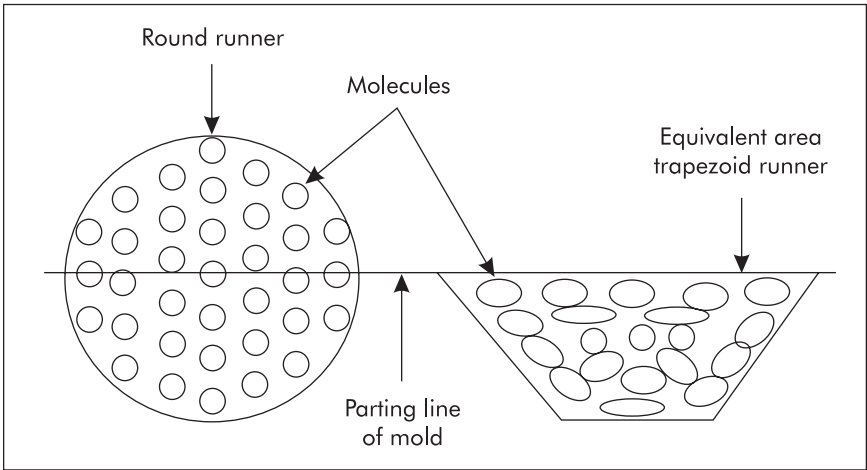


Figure 16-7. Comparing round runner cross-section with trapezoid.

be able to fill every available section of the cavity image and duplicate that image exactly.

If the trapped air is not allowed to escape, it becomes compressed by the pressure of the incoming material and is squeezed into the corners of the cavity, where it prevents filling and causes defects. The air can become so compressed that it ignites and burns, consuming available oxygen and charring the surrounding plastic material.

The most efficient method of allowing the trapped air to escape is to grind air vents into the parting line of the mold. Proper vent design is discussed in detail in Volume I of this series, but suffice it to say that there cannot be too much venting. As long as the vents are of the proper thickness and length, they can be any width, and they can be any number. A good rule of thumb is to allow at least 30% of the parting line perimeter for venting, as shown in Figure 16-9.

Measuring the parting line perimeter of this mold shows that there is a total of 10 in. (254 mm) of parting line ($4+4+1+1$). Using 30% of that figure as the recommended minimum for venting gives a total of 3 in. (76 mm) (30% of 10 in.) that should be used for venting. By spacing 1/4-in. (6.4-mm)-wide vents equally along that parting line, there is a total of 12 vents. Common sense dictates that there should be vents in corners opposite the gate, and the others are simply spaced equally from there. The vents could be 1/2 in. (13 mm) wide, instead of 1/4 in. In that case there would be a total of six vents equally spaced. They could be 1 in. (25.4 mm) wide, which means there would be a total of three vents. As long as they total 30% minimum of the length of the parting line perimeter, there will be enough venting. By using vents that are

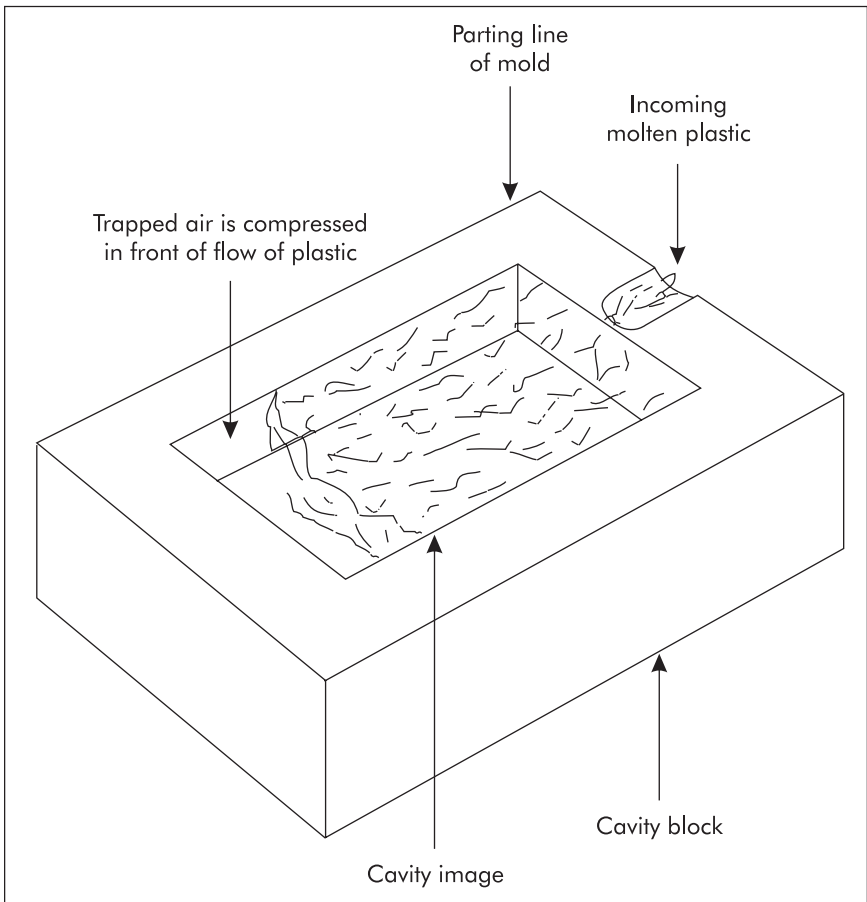


Figure 16-8. Air trapped by incoming flow.

only 1/8 to 1/4 in. (3.2 to 6.4 mm) wide, more vents can be utilized and trapped air will have a chance to escape the mold more effectively.

It is also a good practice to vent the runner. This eliminates much of the air that is trapped in the runner path from being pushed into the cavity in the first place. A 1/4-in.-wide vent placed every inch (25.4 mm) along the runner path is adequate. These vents can be staggered so both sides of the runner path are vented, as shown in Figure 16-10.

We have looked at both the injection and ejection halves of the mold. We have seen how the plastic material gets into the mold, and how the finished product gets out of the mold. Now we examine some other items of consideration regarding molds.

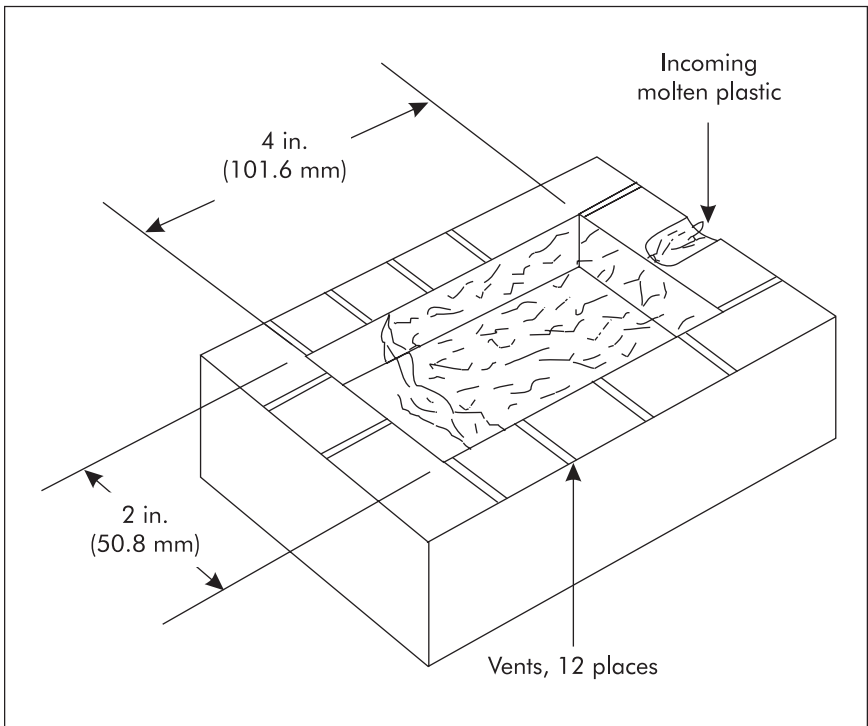


Figure 16-9. Venting of parting line.

Typical Tool Life

The actual usable lifetime of a mold is dependent on a variety of things, primarily the anticipated annual product volume requirement. The higher the volume, the more robust construction needed for the mold, therefore the longer it should last. However, just as with any other high-cost item such as a car or house, once the anticipated use is achieved, the mold is not ready to throw in the trash. It can be recycled for another product.

When designing and building a mold, one of the first items to consider is the material used to build the mold. The majority of molds are made of varying grades of steel. The better grades are used for longer life and less maintenance. But some molds are made of aluminum and some of epoxy. Others are made of copper, and some are made with combinations of some or all of these materials. The primary difference among these molds is the maintenance factor. The better grade steel molds will require less maintenance and repair than the lower grade materials, all other things being equal. However, all the molds can have equal lifetimes if they are properly maintained and repaired as required. The higher grade

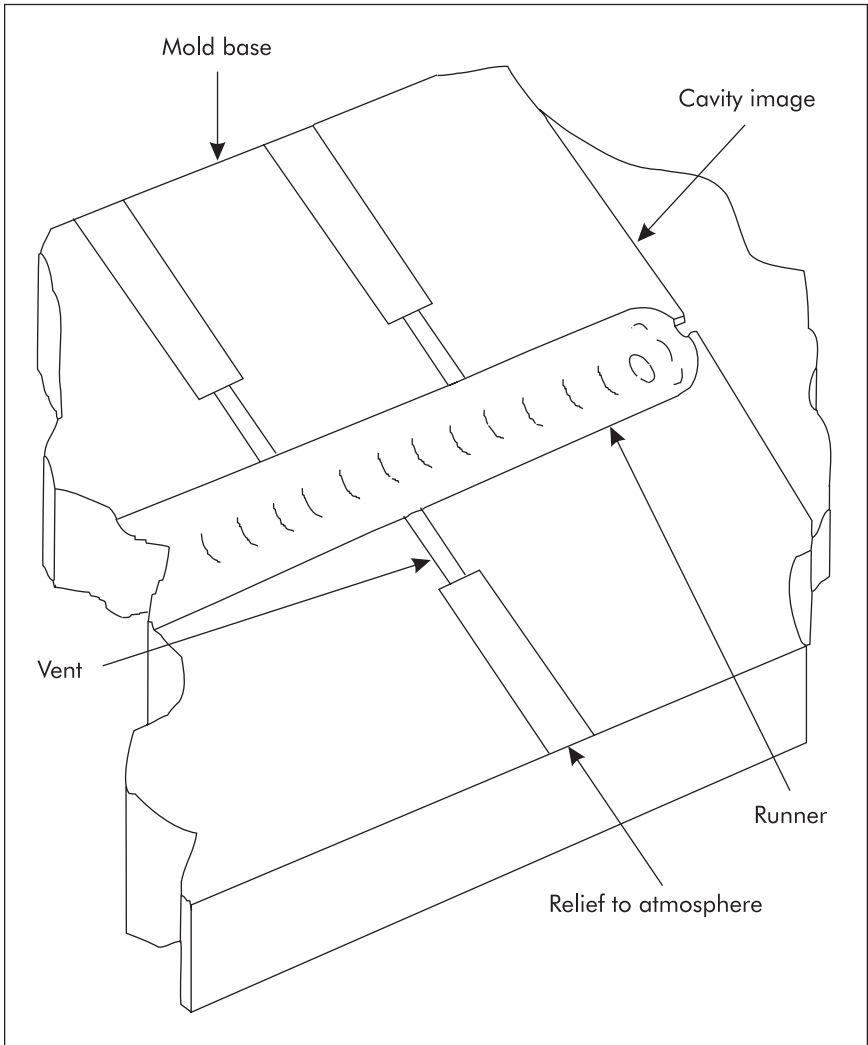


Figure 16-10. Venting of the runner.

molds will cost more to build but will require less maintenance. Lower grade molds will cost less to build but require more maintenance.

Generally speaking, high-volume production molds are expected to last approximately 10 years. Some low-volume molds are expected to last only a year or less. Therefore, to keep investment costs reasonable, the lower-volume molds are usually built to lower standards than the higher-volume molds. Obviously, it will cost much more to maintain or repair them to last longer than their intended

life. A decision must be made early in the development phase as to the expected annual volume and product life to determine what level of quality will be required in the production mold. These decisions should be shared with the appropriate vendors.

Ownership of the Mold (and Other Tools)

Generally, a mold and other sets of tooling are designed and built for a specific product, intended for only that product. Therefore, it is generally accepted that the customer who owns the product design also owns any associated tooling. Payment for that tooling is usually a separate item from piece part payment, but sometimes the customer requests that tooling costs be amortized over the number of pieces manufactured from that tooling. This is risky for the vendor using or building that tooling because the number of parts produced may never approach the intended numbers sufficient to cover the actual costs. The more accepted practice is to have the customer pay for the tooling up front, separate from the cost of unit production.

The customer accepts ownership of the tooling when first article samples are accepted. These must be accepted prior to expecting the molder to mold production quantities of parts. The molder is expected to pay for routine maintenance and repair of the mold while it is in the molder's possession, for the agreed-upon expected life of the mold. Engineering changes and design modifications by the customer must be paid for by the customer. Once the product life ends, or the customer decides to place production orders elsewhere, the vendor must send the mold, in usable condition, to the customer. The customer can then do what he or she wishes with that mold. In most states of the U.S., the vendor has the right to keep the mold (or other tooling) until all monies owed by the customer are paid.

There are some cases where the customer owns only a portion of the tooling required to produce his or her parts. An example of this situation is when universal tooling is used. These tools consist only of interchangeable sets that fit within universal frames or housings owned by the vendor. If the tooling is to be run at another vendor, that other vendor must also possess a frame or housing that is a duplicate of that for which the tooling was built. Otherwise another set of tools will be needed or another frame will have to be purchased. If it is of a commercial nature it will mean buying it from the manufacturer. But if the design is a proprietary one, owned by the first vendor, it may not be copied, and new tooling will no doubt be required. These issues should be reconciled and understood before placing orders with any vendor.

What to Expect from Molds

The first item to consider in determining mold design and construction is how many cavities to make. A single cavity, with an average 30-second cycle time, is

capable of producing 120 pieces per hour, neglecting scrap (5%) and setup times. If the mold runs 24 hours a day for 52 weeks (7 days per week), the maximum number of pieces possible is 1,048,320 annually. A captive molding operation likes to run on this basis. But most custom molding shops base production on 5-day weeks, 50 weeks a year, and 24 hours per day. And they do not wish to run a single mold more than about 3 to 4 months a year. If we use 3 months as the maximum, a single cavity mold at a custom molder would then be capable of producing only 180,000 pieces annually. If the customer wished to have 300,000 pieces molded, the molder would probably suggest a two-cavity mold, which would make each piece less expensive.

Molds are built to standards that vary depending on what is expected from the mold. Usually, the mold designer or moldmaker will make that determination based on the product design requirements and estimated annual volumes projected, as well as the anticipated life of the product program as dictated by the customer. The customer, however, has the right to request any type of mold he or she wants, and may override the decision of the vendor. Of course there is a cost associated with the type of mold selected.

The highest quality mold will be made of a high-grade tool steel, chrome- or nickel-plated to combat wear. It will be constructed using laminar technology rather than having the entire cavity image cut into a solid block of metal. Laminar construction is that which uses individual components to make up the cavity image, much like a jigsaw puzzle. These components are then mounted into a block which, in turn, is mounted into a “pocket” in the mold base. This method of construction, while expensive, allows for easy repair or alterations specified by engineering changes. If long life is expected, high volumes are predicted, and a quality mold is desired, this is the type to request.

If cost is the top priority issue, shortcuts can be taken to reduce the cost (thus quality) of a mold. For instance, lower-grade, less-expensive steels can be used. In fact, even aluminum can be used if certain limitations are accounted for. And the cavities can be cut in the “solid” rather than using the more expensive laminar method. While this process is less expensive, it does present a high cost potential when repairs or engineering changes are required.

The average mold cost today is approximately \$35,000. A high-quality version of that same mold might run to \$55,000, while a low-quality version might run as low as \$20,000. While the low cost may be an appealing inducement initially, you *do* get what you pay for. If the mold must last more than 2 years, a shift to higher quality is probably justified. And if the mold is to last 3 years or more, that shift becomes almost mandatory. An axiom in the business is that a mold has a total, definite cost associated with it. That cost is spread over the entire life of the mold and can be paid with a larger portion up front (initial cost), with the balance being presented in low installments for repairs and

changes. Or the up front cost can be low, with the balance being presented in high installments for repairs and changes.

Those charged with purchasing a mold should be aware that moldmakers may purposely quote a low figure to get the work. This may happen with present vendors as well as new vendors. This practice should be discouraged by the buyer because it presents the potential for getting a lower quality mold than that desired. On the other hand, if a moldmaker is busy and doesn't need the work, he or she may quote very high instead of not quoting at all. If he or she gets the work, justification to add overtime or hire more people to build the mold will be provided. If the moldmaker doesn't get the work, he or she is no worse off than before. To assure that a good quality mold is being purchased at the optimum price, a mold buyer can request quotes from six to 10 moldmakers. Then the low-cost vendor and the high-cost vendor can be contacted and asked if they wish to reconsider their quotes. A reason should not be given for the buyer's question. It is possible that the moldmakers misquoted due to math errors, and they should be given the chance to re-quote. Once the final numbers are in, the buyer can throw out the low and high quotes and go with one of the middle-cost vendors. While this practice does not guarantee quality of the mold, it does create the highest possibility that the quality will match the price.

Before making a commitment, visit (or survey) the potential moldmaker (or molder) vendor to ensure that they are capable of producing a mold or product that will meet your requirements. Of course the person making the visit must have the expertise required to make this determination.

And, finally, you, as the customer buying the mold, must understand that the more that is required from the mold, the higher the cost and the longer it will take to build. Tight tolerances are much more costly than standard tolerances. Fancy textures are more costly than simple textures. Plating is an additional cost for any mold. Sometimes these are all necessary, but if that is the situation do not expect the mold to be low-cost.

SUMMARY

The injection mold is the heart of the injection molding operation. The plastic molded product is formed within the mold, in cavity images machined into the "A" and "B" plates.

The injection half of the mold is also called the "A" half and, in addition to cavity images, contains the locating ring and sprue bushing. These are used to align the mold with the injection barrel of the machine and guide the plastic into the mold.

Hot runner systems are used to eliminate sprues and reduce cycle times.

The clamp half of the mold is also called the “B” half and, in addition to cavity images, contains the ejection system. Typically, a knockout rod connected to the machine is used to actuate the ejector system of the mold.

Critical areas of concern regarding mold design are gate location, runner design, and venting concepts.

High-quality molds cost more and take longer to build than low-quality molds. Request only the highest quality necessary for a specific product and program life.

QUESTIONS

1. Why is a mold needed for the injection molding process?
2. How would you define the purpose of the sprue bushing?
3. Describe “flash” and list two causes of it.
4. What are the two major advantages of using hot runner systems?
5. Name the three parts of an ejector pin.
6. How is the ejector system typically actuated?
7. Where should the gate be located if at all possible?
8. Which shape is best for the cross-section of a conventional runner?
9. What causes air to be trapped in a mold?
10. What can be done to a mold to allow trapped air to escape?
11. Why should the runner be vented?

Understanding Defects

OVERVIEW

While the product designer or material engineer is not often involved with troubleshooting defective plastic parts, we include this chapter because it explains the primary causes of some common defects and how the troubleshooter can determine what must be done to eliminate them.

Too often, when technicians, engineers, or operators are presented with a part (or set of parts) with defects deriving from molding problems, they start turning dials, flipping switches, and adjusting timers without understanding what they are doing or knowing what results to expect. This is a common occurrence and has its genesis in the practice of troubleshooting a defective part by way of doing something (anything) that worked in the past when a quick fix was desired (but not possible). It doesn't have to be that way. The situation should be such that the troubleshooter can objectively analyze a molding defect and eventually determine a possible solution. The solution should be attempted, followed by another decision. If the first solution doesn't work, another solution should be devised and attempted. But each solution should be determined independently and rationally. There should be no guesswork, and assistance from outside sources should be welcomed and pursued if necessary.

One common source of troubleshooting assistance comes from the material suppliers. They usually have available a convenient laminated guidesheet that tells what to do if certain defects are encountered. These guidesheets are well researched and a troubleshooter may eventually find the answer to a specific problem but may never know what caused the problem or why the particular solution actually worked. The guides don't go into that kind of detail. If they did they would be written like a book. The Society of Manufacturing Engineers (SME) has a three-tape video set (titled *Troubleshooting Injection Molding Problems*) that graphically demonstrates the causes and solutions to the most common defects.

The best approach to troubleshooting is to use as many of the above sources as possible, and mix in a good dose of common sense.

WHAT CAUSES DEFECTS?

A study conducted over a 30-year span (1963 to 1993) by Texas Plastic Technologies* analyzed the root causes of most common injection molding defects. Because troubleshooting takes effect after acceptable parts have been molded, the study only investigated defects that were process-related and did not include those resulting from poor basic product design. It was determined that the defects could be traced to problems with one or more of the following items: the molding machine, the mold, the plastic material, and the molding machine operator. Most interesting was the percentage each of these items contributed toward the cause of the defects. Figure 17-1 summarizes the results.

Many of us in the industry believe that the most frequent cause of defects is the material, with the operator coming in a close second. But as Figure 17-1 shows, the most frequent cause of defects is the molding machine. Thus, when troubleshooting, the first place to look for a solution to a defect problem is the machine, because the answer will be there six out of 10 times.

A troubleshooter must be able to approach a problem with an objective mind. What solved a problem one day may not solve the same problem another day. Because of the large number of parameters and the variables of these parameters, and the way they all interact, many solutions may exist for a single problem. Likewise, many problems may be fixed using a single solution. So the troubleshooter must think through the problem and make sure the proper solution is chosen. This is done by applying objectivity, simple analysis, and common sense.

The first step is to visualize the way a process should be running. Most troubleshooting is actually performed after a specific job has been running successfully for an extended period of time. There has been an initial setup and debugging process and the mold has been accepted for production. Then, after running successfully, parts begin to be molded with defects. This is when the troubleshooter is brought into the picture. This is also when common sense and objectivity must be brought into play.

Visualizing what happens to the plastic as it travels from the hopper through the heating cylinder and through the flow path to the cavity image, you can determine what may have changed to cause defects. A heater band could be burned out, or an injection pressure valve spring may be weak, or cooling water lines may have become blocked. Any of these will cause specific things to happen. A thorough understanding of the molding process will help determine the cause.

On the following pages are listed 24 of the most common molding defects. There is also a listing of the most common cause of each defect along with the

*See Chapter 14, page 300.

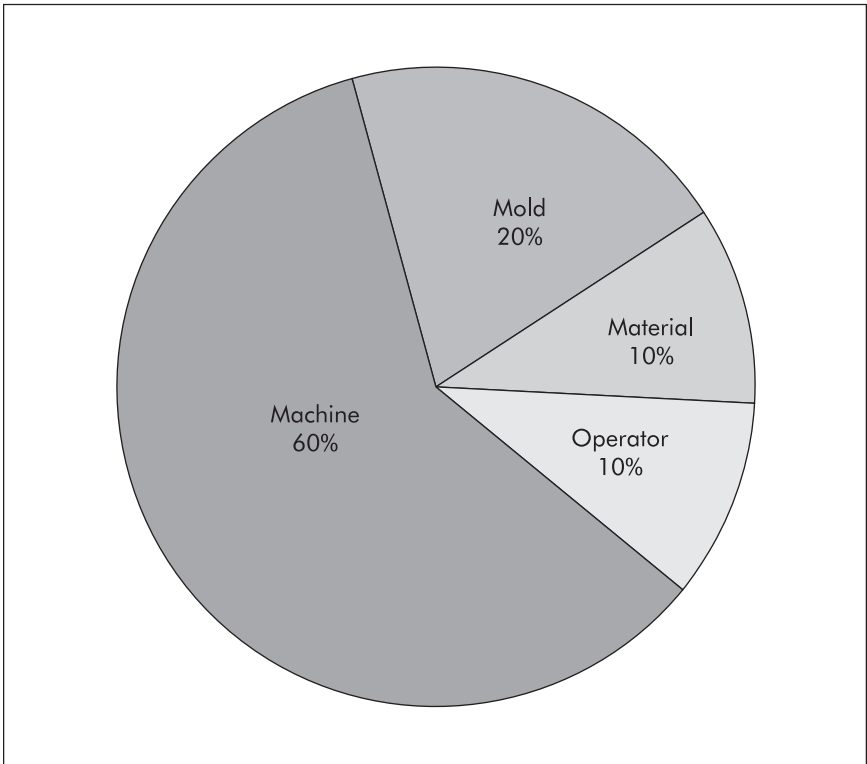


Figure 17-1. Distribution of defect causes.

most popular remedy suggestions. The list of defects is not intended to be all inclusive, nor is the list of causes and remedies. This section is only intended to assist the product designer or materials engineer to understand some of the potential defects that may show up in their molded parts.

COMMON DEFECTS AND REMEDIES

Black Specks or Streaks

Excessive Residence Time in Barrel

Between 20 and 80% of the barrel capacity should be injected each cycle. If the plastic stays in the barrel longer than normal it will begin to degrade. This degradation results in carbonized plastic which appears as small black clusters. These can be carried through the melt stream and show up as spots or streaks in the molded part, visible on the surface of an opaque part and throughout a transparent part. The solution is to place the mold in a properly-sized machine.

Contaminated Raw Material

Contamination can be the result of dirty regrind, mixed regrind, improperly cleaned hoppers or grinders, open or uncovered material containers, and even poor quality virgin material as supplied by the manufacturer. The remedies include dealing with high-quality suppliers, using good housekeeping practices, and properly training material handling personnel.

Blisters

Back Pressure Too Low

As the material is heated and augered through the heating cylinder, air becomes trapped within the melt. One of the uses of back pressure is to force this air out before it gets injected into the mold cavity image. Back pressure should be set at 50 psi (345 kPa) and increased in increments of 10 psi (69 kPa) until the ideal setting is reached, but never beyond 300 psi (2,068 kPa).

Blush

Mold Temperature Too Low

If the mold is too cold, the flow of the molten material is hindered and the material solidifies before it fills and packs the mold. Blush (dull finish) will appear in the last area to be packed, usually the gate. Increasing the mold temperature allows the material to flow farther and pack properly.

Bowing

Clamp Opens Too Quickly

To increase the number of cycles produced in an hour, molders may sometimes increase the speed at which the clamp opens the mold at the end of the molding cycle. If this is done at the very instant of the “mold open” portion of the cycle, there will be a tendency for the part to hang up on the injection half (dead half) of the mold. As the mold continues to open, the part will snap back onto the clamp half (live half) of the mold and the result may be a bowed part from this distortion. The solution is to make sure that the first 1/4 in. (6.35 mm) of “mold open” is set at a slow speed. The balance of the opening cycle can then be set at a much faster speed.

Improper Handling

It is possible that the operator has improperly handled molded parts after ejection from the mold. If parts are packed for shipment too soon after molding, the heat that is retained in them may not be allowed to dissipate properly and they could take a bowed set. Also, relief operators may not handle the parts the same way the main operator does. Proper instruction is required to remedy that situation, and analysis of the packaging process should be performed to minimize the problem.

Brittleness

Excessive Moisture

All materials need a small amount of moisture to properly process, usually in the area of 1/10 of 1%. Some materials such as nylon and ABS are hygroscopic by nature and readily absorb moisture from the atmosphere, even after initial drying. These are difficult materials to keep dry. Moisture causes brittleness because the water droplets turn to steam when heated in the injection unit and this steam explodes through the melt stream, causing voided areas. These voided areas are not properly bonded and easily break apart when they are subjected to any mechanical forces after molding.

Some materials (especially hygroscopics) may require conditioning after molding to replace the moisture removed in the molding process. Nylons, for example, normally must be conditioned by either annealing in 300° F (149° C) glycerin for 4 hours, or being placed for 4 days in sealed bags filled with water. Without this conditioning, the plastic will be brittle, the direct result of using *proper* drying procedures to mold the plastic.

Bubbles (Voids)

Section Thickness Too Great

When a plastic part consists of varied wall thicknesses (instead of one steady thickness) the thicker walls will cool (and solidify) last. There will be a pressure loss in those thick areas as they continue to cool after the thinner areas have solidified. The plastic will pull away toward the solid section and cause a void in the thick section. When the void is on the surface of a part, it appears as a sink mark. When it is below the surface it appears as a bubble. The best (although expensive) solution is to use metal coreouts to thin the thicker wall. Or, if possible, change the wall thickness so that the thicker section is no more than 25% thicker than the thin section. This will minimize the void.

Excessive Moisture

Excessive moisture can get trapped in the resin as the molding process progresses and show up as bubbles in the molded part. The moisture actually turns to steam during the heating process and cannot escape from the material, so it forms a gas pocket which shows up as a void. The obvious solution is to properly dry the material before molding.

Burn Marks

Improper Venting

Venting systems are placed in molds to exhaust any gases or trapped air that might be present. If the vents are not deep enough or wide enough, or if there are not enough vents, the air is compressed before it is all exhausted. It then ignites and burns. Vents must be a minimum of 1/8 in. (3.18 mm) wide. The vent

land should not be more than 1/8 in. long. Blind areas, such as the bottoms of holes, should incorporate vents machined on the side of ejector pins located there. There should be enough vents on the parting line to equal 30% of the distance of the parting line perimeter. Thus, a 10-in. (254-mm)-long parting line perimeter would have 12 vents, each of which measure 1/4 in. (6.35 mm) wide (3 in. [76 mm] total).

Excessive Regrind

The use of regrind may have to be limited, especially when using heat-sensitive materials such as PVC. Regrind material tends to absorb heat in the injection barrel at a slower rate than virgin, due to the irregular surfaces and larger size of the regrind particles. This results in a longer heating cycle, which causes the virgin pellets to overheat and degrade. The degradation takes the form of burned particles which are transported through the melt stream into the cavity. Limit regrind use to no more than 5 or 10%. If the shot size is small (less than 20% of barrel volume) it may require no regrind at all. A possibility is to start with all virgin and slowly build up regrind use by salting in regrind at 2% increments until burning occurs. Then drop back one 2% step and use the resultant ratio for future molding.

Clear Spots

Barrel Temperature Too Low

Low barrel temperatures result in an improper blending of molecules due to unmelted particles. These particles travel through the melt stream and enter the cavity. In transparent parts they show up as clear spots, but even in opaque parts they may show if near the surface. Cutting open an opaque part will reveal the spots as voids. Increasing barrel temperature will reduce the tendency for unmelted particles. Increase in 10° F (6° C) increments and allow the temperatures to stabilize (10 cycles) before increasing again.

Inconsistent Cycles

Erratic cycling of the machine can result in erratic heating in the barrel, causing hot spots and cold spots. Material from the cold spots may continue through the melt stream without being properly heated and will show up as clear spots in transparent parts. Training the operators and explaining the importance of consistent cycles should be accompanied by examples of defective parts run during inconsistent cycling.

Cloudy Appearance

Uneven Packing of Cavity

Cloudiness caused by uneven packing can normally be traced to improper gate or runner sizing or location. The material enters the cavity at the wrong spot and does not pack properly against the mold steel in all areas. The material

solidifies without replicating the mold finish, and this appears as a cloudy area. There is also the possibility that one area of the molding surfaces was not polished as well as the other areas, resulting in a cloudy appearance in that area. Make sure that the mold is properly polished. When it is, use a computer program to determine the proper gate size, number, and location for a specific product design. If this is not available, it is possible to determine proper parameters through trial and error, but it is very time consuming and expensive. A design consultant may be a good investment.

Contamination

Improper Regrind

Regrind may contain numerous contaminants such as food containers, soft drink spills, dust and dirt particles, and other plastic materials. This is usually due to poor housekeeping practice and careless material handling procedures. This contamination can be greatly reduced by proper instruction of personnel, highly visible labeling of regrind containers, proper labeling of trash containers to differentiate them from material containers, tight fitting covers for regrind (and any plastic) material containers, proper cleaning of regrind machines, and care during material changeovers.

Poor Housekeeping

An operator can cause contamination through actions such as eating at the molding machine station. Potato chip salt and soft drink spills are common sources of material contamination. Dust from sweeping can enter the hopper if it is not covered. And, in rare cases, operators have been known to “force a break” by intentionally throwing trash in the hopper. Operators should be instructed on the importance of maintaining good housekeeping practices and supervisors should be held to the same standards.

Cracking

Molded-in Stresses

Stresses can be molded into a product through the molding machine by excessive packing or too fast a filling rate. The plastic is injected and held against the restraining surfaces of the mold cavities. When the part is ejected, the cooling process continues (for up to 30 days) and the highly pressured plastic may begin to relieve. If the skin of the molded part is not yet solid enough it will split open in the form of cracks. Reduce the injection pressure and speed to the lowest numbers that will successfully mold the part. This reduces the tendency to mold in stress.

Insufficient Draft or Polish

Draft angles should be an absolute minimum of 1° per side to facilitate easy removal of the part from the mold. Ejector pressure may cause the parts to crack if less than 1° per side is used. Also, rough cavity surfaces (and other undercuts)

cause a drag on the part as it ejects. This may cause cracking if the ejection pressure is increased to push the part over this rough surface. Cavity surfaces should receive a high polish when the mold is built and should be repolished as the need arises.

Crazing

Because crazing is simply a very fine network of cracks, the same causes and remedies apply that are mentioned under “Cracking,” in the previous example.

Delamination

Foreign Materials or Additives

If a pigment is being used to color the resin, it may not be compatible, such as in the case where soap is used in the manufacturing of the pigment. If a color concentrate is used to color the resin, it must be of a material compatible with the base resin. And, if accidental mixing of two incompatible resins has occurred, they will not bond. In all these cases, nonbonding of the materials used will result in delamination of the molded layers of the finished product. Check with suppliers of any additives to make sure the proper grade is being used. Also, ensure that all materials are properly identified to ensure that incompatible materials are not being mixed.

Excessive Mold Release

If a mold release is required at all, it is necessary to limit its use. Too much mold release will cause a penetration of molded layers by the mold release itself. This will keep the layers from bonding and delamination will result. Keep mold release away from presses unless absolutely necessary, and then use it only as a quick fix until the cause of the sticking can be rectified.

Discoloration

Improper Mold Temperature

In general, a hot mold will cause the material to stay molten longer and allow the molecules to pack tighter. This results in a very dense part which appears darker due to that density. On the other hand, a cold mold will cause a loss in gloss because the material cools before it can be forced against the mold surface, and this will cause a less dense part which will appear lighter. The mold temperature should be adjusted, within the limits set for the particular plastic, to the point at which the material has the proper shade and gloss properties. Be aware that the color will change slightly after the part has been completely cooled.

Contamination

Material may appear discolored if it is contaminated with any number of items including wrong regrind, fabric strands, thermally degraded material, and food

particles. Also, if the entire shot of material has been exposed to excessive temperatures, it will be darkened. Proper housekeeping will minimize most of this type of discoloration, and closer control of proper molding temperatures will minimize the rest.

Flash

Excessive Injection Pressure

It's possible that too much injection pressure will partially overcome the clamp pressure of the machine and cause the mold to open slightly during the injection phase. If this happens, a small amount of plastic actually seeps out of the mold. This seepage is called flash. Also, excessive pressure may force plastic into the clearance hole around ejector and core pins. This is also flash. Reducing the injection pressure minimizes flashing conditions. If the mold design allows, begin the molding process with very low pressure and slowly increase from shot to shot until the cavities are filling properly. This should be done in 50-psi (345-kPa) increments when the mold is almost filled.

Improper Flow Rate

Resin manufacturers supply materials in a variety of flow rates. Thin-walled parts may require easy-flow materials while thick-walled parts may use stiffer materials. The stiffer materials are usually stronger. If a fast-flowing material is used, it may creep into small crevices where thick materials could not. Flash could be the result. But even with thicker materials, if the flow rate changes to slightly thicker yet, more pressure may be required to inject the material and this could blow the mold open, also causing flash. Use a material that has the stiffest flow possible without causing nonfill or flashing in the mold. This can be pre-determined somewhat by consulting with a materials supplier.

Flow Lines

Inadequate Injection Pressure

Flow lines may be the result of improperly bonded material. If injection pressure is too low the tongues of material that enter the cavity are not packed together to form smooth layers against the molding surface. The material actually starts to wrinkle as one layer tries to crawl over the already cooling layer outside of it. Increasing the injection pressure will force the layers together quickly while they are still hot enough to bond tightly.

Gloss (Low)

Inadequate Injection Pressure

If there is not enough injection pressure, the molten material is allowed to cool and solidify before it has had a chance to pack and force itself against the molding

surfaces of the cavities. If this happens, the material will not reproduce the finish on the mold and will simply cool down with no pressure against it. This results in a dull finish on the product. Increasing the injection pressure will force the material against the mold surface, minimize flow lines, and duplicate the finish and gloss of the mold.

Mold Temperature Too Low

Generally, a hot mold produces a higher surface gloss on a molded product than a cold mold. This is because the particles are allowed to stay molten longer, which results in a dense product. Also, the material is able to duplicate the surface of the mold steel better because it is so dense. Increase the mold temperature to the point at which the material has the proper flow and packs the mold. This will result in higher gloss. Conversely, if too high a gloss is produced, decreasing the mold temperature will reduce the gloss.

Jetting

Improper Gate Design or Location

If material is injected directly across a flat cavity surface it tends to slow down quickly as a result of frictional drag and cools before the cavity is properly filled and packed out. When this happens, flow streams tend to form and the molded part surface has the familiar jetting appearance. Relocate or redesign the gate so that the molten plastic is directed against a metal surface instead of across a flat surface. This will cause the material to disperse and continue to flow instead of slowing down.

Knit Lines (Weld Lines)

Barrel Temperature Too Low

Knit lines are the result of a flow front of material being injected at an obstruction in the mold cavity. The flow front breaks up into two separate fronts and goes around the obstruction. When the two fronts meet on the other side, they try to weld back together again (knit) and form a single front once more. If the barrel temperature is too low, the material does not keep its heat long enough and the two fronts cannot make a good knit because the material has begun to solidify. Increasing the barrel temperature will allow the melt fronts to stay molten longer and knit properly.

Nonfill (Short Shots)

Insufficient Material Feed

The most common cause of nonfill is not enough material prepared in advance for injection into the mold. Increase the amount of material being fed to the mold by adjusting the return stroke of the injection screw so that more material is transferred from the hopper system with each rotation of the screw. Adjust

this setting until there is between a 1/8- and 1/4-in. (3.18- and 6.35-mm) cushion at the front of the injection cylinder.

Insufficient Venting

Venting is used to remove trapped air from the closed mold so molten material will be able to flow into every section of the mold. If the air is not removed, it acts as a barrier to the incoming plastic and will not allow it to fill all sections of the mold. The result is nonfill. The mold should be vented even before the first shot is made. Vent the runner first, and then create enough vents on the parting line to equal 30% of the length of the perimeter surrounding the cavity image. An additional approach is to use a vacuum system in the mold to help pull the trapped air out before injecting material.

Shrinkage (Excessive)

Mold Temperature Too High

Generally, a hot mold causes the material to stay molten longer, which may result in the required skin not properly forming before ejection of the product. When this occurs, the still-cooling material continues to shrink because there is no normal restraining skin to hold it from shrinking too much, and the product will shrink beyond the normal dimensions. Decrease the mold temperature until the material maintains the proper flow and fills the cavity without shorting. This should be done in 10° F (6° C) increments and once the desired level is reached a single increase of 10° F should be added to compensate for fluctuation in the temperature control units.

Early Gate Opening

If the operator is speeding up the cycle by opening the gate too soon, the cooling plastic may not have had a chance to solidify enough to form a proper skin on the molded product. If this skin is not solid, the remaining plastic that cools pulls the skin with it and, with nothing restraining it, it continues to shrink beyond expected rates. Instruct the operator that cycles that are too fast may cause defects in the molded product that are not even visible.

Sink Marks

Excessive Rib Thickness

Ribs are normally designed into a part to add strength in a given area. If the rib thickness is the same as the adjoining wall thickness, an excessively thick area is created at the junction of the rib and the wall. This thicker area takes longer to cool and as it does it pulls in the already cooled and solidified area around it, resulting in a sink mark. The rib wall should be designed to be no more than 60% of the adjoining part wall. Thus, if the part wall is nominally 0.090 in. (2.29 mm), the thickness of the rib should not exceed 0.054 in. (1.37 mm). This

keeps the junction area relatively thin so it will cool at the same rate as the surrounding areas and minimize (or even eliminate) sink marks.

Splay (Silver Streaking)

Excessive Moisture

If the material was not properly dried, the excessive moisture will turn to steam as it travels through the heating barrel. This steam becomes trapped and is carried into the mold cavity where it is usually forced to the surface and spread across the molding surface of the cavity. It appears as streaks of silvered char, which is splay. Make sure all materials are properly dried. Even materials that are not hygroscopic (such as nylon) must have surface moisture removed before molding. Drying conditions are critical and the material suppliers have documented conditions for specific materials and grades.

Warpage

Inadequate Injection Pressure or Time

If too little injection pressure or time is used, the plastic material will tend to cool down and solidify before the mold is packed out. Then the individual molecules of the plastic are not packed together and have space to move into as the part is cooled. They relax during the cooling period and are allowed to move about. While the outer skin of the product may be solid, the internal sections are still cooling and the movement of molecules here determines the degree of warpage. Increase injection pressure or time to contain the cooling molecules in a rigid position (packed) until they are solid enough to prevent movement.

TROUBLESHOOTING TIPS

Successful troubleshooting usually requires making changes to an existing process. These changes will sometimes have an immediate effect, but in all cases any changes will also have long-term effects. This is because the total molding process requires a certain amount of time to stabilize after any change is made. For instance, an increase in barrel temperature will alter the flow rate of a material after only a few minutes, but that increase also has an effect on the injection speed after a few more minutes because the material is easier to inject. A faster injection speed may initiate a tendency for flash to begin after a few more minutes. There are two principal rules to follow when making adjustments to molding parameters:

1. Make only one change at any time.
2. Allow the machine to stabilize for a period of 10 to 20 cycles after any single change is made to the process.

Troubleshooting can be a time-consuming process because of the amount of time required to allow the machine to stabilize between changes. However, without that stabilizing time, so many changes can be made that no one could determine which change actually solved the original problem. Of course the major concern is that, when many changes are made at once, the entire process quickly goes out of control and runs in total confusion. So a troubleshooter must be objective in analysis, selective in solution, and most of all, patient in activity.

SUMMARY

Troubleshooting is a process that should be performed with objectivity and patience.

Common sense should prevail when troubleshooting. Random efforts at finding solutions will only complicate matters and create an out-of-control situation.

Selected parameter changes should be performed only one at a time and the process should be allowed to stabilize before any additional changes are made.

The primary culprit of molding defects is the molding machine (60% of the time) followed by the mold (20%), the raw material (10%), and the operator (10%). Unfortunately, the operator is usually the first to be blamed for defective parts.

Observation is the first tool the troubleshooter should utilize. By visualizing what happens to the plastic as it travels from the hopper through the heating cylinder and through the flow path to the cavity image, the molder can determine what may have changed that could be causing defects.

QUESTIONS

1. Not considering initial product design, what are the four root causes of most injection-molded defects?
2. What percentage of the total are each of the four causes mentioned above responsible for?
3. What percentage (range) of the barrel capacity should be emptied during every injection cycle?
4. What is the term used to describe a material that absorbs moisture from the atmosphere?
5. What happens to moisture in the material as it travels through the heating cylinder of the machine?

Answers to Chapter Questions

CHAPTER 1

1. 1868.
2. 1872.
3. World War II created a demand for inexpensive, mass-produced products. New materials were invented for the process on a regular basis and technical advances resulted in increasingly successful applications.
4. 1979.
5. Injection molding can be defined as a process consisting of heating a plastic material to a point at which it becomes soft enough to force into a closed mold, at which point the material cools enough to solidify and form a product.
6. Over 100.
7. Four. These are: temperature, pressure, time, and distance.
8. 500° F (260° C).
9. The softening (or melting) of the plastic is achieved by causing the individual molecules within the plastic material to go into motion. This is accomplished by applying heat to the molecules. (*Note:* Both parts of the answer must be present to obtain full credit.)
10. Front, center (middle), and rear.
11. 225° F (107° C).
12. A heat exchanger.
13. Injection pressure, holding pressure, and clamping pressure.
14. The melt index test is a good reference test to determine the relative thickness (viscosity) of a plastic material, thereby its relative ease of flow during the injection process.
15. Clamp pressure can be defined as the amount of pressure required to hold the mold closed against injection pressure.

CHAPTER 2

1. Any complex, organic, polymerized compound capable of being shaped or formed.
2. A reaction caused by combining a monomer with a catalyst, under pressure, with heat.
3. Crude oil and natural gas.
4. It is necessary to react monomers of different groups together.
5. An alloy must be homogenous while a blend is not.
6. A plastic material which when heated, undergoes a *physical* change. It can be reheated and reformed over and over again.
7. A plastic material, which when heated, undergoes a *chemical* change and “cures.” It cannot be reformed, and reheating only degrades it.
8. Amorphous and crystalline.
9. Amorphous materials are those in which the molecular chain structure is random and becomes mobile over a wide temperature range.
10. Crystalline materials are those in which the molecular chain structure is well ordered and becomes mobile only after being heated to its melting point.
11. Any three of the following: clear, low shrinkage, softens (no melt), high impact, poor chemical resistance, poor lubricity.
12. Any three of the following: opaque, high shrinkage, melts (no soften), low impact, good chemical resistance, good lubricity.

CHAPTER 3

1. Reinforcements and fillers.
2. Reinforcements are used to enhance physical strength properties.
3. Fillers are used to enhance properties other than strength.
4. Fiberglass.
5. The weight will be increased.
6. Lower.
7. A special class of aromatic polyamide fibers developed by DuPont.
8. Edison used a carbonized fiber (sewing thread) for the first commercial glass lighting lamp.
9. Aluminum, silica, quartz, and zirconia.
10. Diatomaceous earth, ceramic oxides, silica.
11. Dyes and pigments.
12. It becomes more flexible.

CHAPTER 4

1. Amorphous and crystalline.
2. Molecular structure.

3. The degree of crystallinity.
4. It gets softer until it is able to flow for molding.
5. Differential Scanning Calorimeter (DSC).
6. By circulating water or oil through cooling channels cut in the mold.
7. 85%.
8. The ordered structure of the crystalline molecules allows the molecules to come in closer contact with each other, thereby taking up less volume than amorphous materials.
9. It takes more energy to break down and re-form the crystalline molecular bonds.
10. They are the lowest of all thermoplastic materials.

CHAPTER 5

1. Physical, mechanical, thermal, electrical, and environmental.
2. Density is defined as a measure of mass per unit volume of any material. It is expressed as pounds per cubic inch or grams per cubic centimeter.
3. Because it implies a time value that is not present.
4. Inch-per-inch.
5. The “constant” in Hooke’s Law is called the modulus of elasticity and it is sometimes referred to as Young’s modulus or tensile modulus.
6. A constant used for determining the stress and deflection properties of plastic structures such as rotating discs or plates.
7. Shear stress = shear load \div area resisting shear.
8. Creep is an increased strain over a period of time in the presence of constant stress occurring from a constant load.
9. Izod impact, Charpy impact, tensile impact, and falling dart impact.
10. It compares the relative ability of various plastics to perform at elevated temperatures while supporting loads.

CHAPTER 6

1. A plastic product being produced at the lowest possible cost with the highest required level of quality.
2. Acrylonitrile-butadiene-styrene.
3. The use of star diagrams allows a visual representation of the strong and weak points of a specific plastic. These show at a glance how one material compares to another.
4. Acetal is known for its ease of machining and resistance to most chemical solvents.
5. TFE, FCTFE (or CTFE), PFEP (or FEP), and PVF.
6. PVF.

7. Thin section stiffness, extreme temperature stability, and excellent chemical resistance.
8. Nylon.
9. From a -275°F to a 375°F (-171°C to a 191°C).
10. Thermoplastic rubbers can be processed without compounding or including additives.

CHAPTER 7

1. Any of the following three (or combination): to determine profitability, to play “what if” games, and to monitor outside vendor costs.
2. Material costs, labor charges, machine hour rate, tooling costs.
3. Because of the difference in specific gravity (therefore weight) of the many available plastics.
4. 100%, because regrind that has not been abused will retain as much as 90% of the properties of virgin.
5. 15%.
6. How much clamp tonnage is required and how big an injection unit is necessary.
7. 20,000 psi.
8. Half (50%).
9. 40%.
10. The Far West and the Northeast.

CHAPTER 8

1. Any operation that is performed on a product after it has been molded.
2. Any two of the following: when volumes are small, if tooling costs are excessive, when time to build the mold jeopardizes marketing schedules, or when a labor-heavy environment exists.
3. 20,000 to 40,000 cycles per second.
4. Any two of the following: polymer structure, melt temperature, stiffness, moisture content, flow rates, mold release agents, plasticizers, flame retardants, regrind, colorants, resin grade, fillers, and reinforcements.
5. A nitrile elastomer.
6. From 1,000 to 5,000 rpm.
7. To ensure adequate bonding of the decorative material to the plastic material.
8. Any two of the following: detergent wash, flame treatment, corona discharge, plasma process, and acid etching.
9. Applied finishes are applied to the product after the molding process, while in-process finishes are placed on the plastic product as a result of (or during) the molding process.

10. Any three of each category as follows:

Applied = painting, plating, vacuum metallizing, hot stamping, pad printing, and screen printing.

In-process = molded-in color, molded-in symbols, two-color (or two-shot) molding, textured surfaces, and in-mold overlays.

CHAPTER 9

1. ASTM (American Society for Testing and Materials) and ISO (International Organization for Standardization).
2. Improper sample preparation can cause improper test results.
3. Any three of the following: dielectric strength, dielectric constant, volume resistivity, surface resistivity, and arc resistance.
4. Any three of the following: shrinkage rate, density, water absorption, moisture content, and melt flow index.
5. Any three of the following: tensile strength, compressive strength, flexural strength, creep, and impact testing.
6. Any three of the following: melting point, heat deflection temperature, Vicat softening temperature, flammability, and limiting oxygen index.
7. Failure analysis (FA) differs from troubleshooting in that FA is usually performed after the product is molded while troubleshooting is usually performed during, or immediately after, the molding process.
8. A resistance to deformation from an applied force.
9. Differential Scanning Calorimeter.
10. Any answer that defines comparing the visible area measured under both peaks for virgin and regrind which are evident on the same scan.
11. Any answer that properly describes comparing the areas under the peaks of individual scans; one for the unknown sample and one for a known, fully-crystallized sample.
12. (Muffle) furnace method and TGA (thermogravimetric analysis).

CHAPTER 10

1. Product design, purchasing, manufacturing engineering, materials engineering, quality, and marketing.
2. Any three of the following: what does the product do?, how will the product be made?, when must production start?, where will it be built?, and how much should it cost?
3. Storage is expensive and should be minimized as much as possible.
4. Just-in-Time.
5. To show the estimated time required to achieve a series of checkpoints along the path of a product development cycle.

6. This is the time at which the product is proposed to team members.
7. During Phase 2.
8. During Phase 7.
9. Identify the needs that require the project in the first place.
10. Any two of the following: uninvolved users, undisciplined management system, ineffective communications, uncontrolled changes, inappropriate technology, and poorly defined products.

CHAPTER 11

1. Defining the product includes activities that place ideal requirements on the product and practical tolerances on those requirements.
2. The seven basic areas of interest to consider when determining the actual requirements of a product to be manufactured are: functional, physical and mechanical, thermal, electrical, flammability rating, weather and exposure limits, and safety and environmental issues.
3. To ensure accuracy, a color chip selection should be requested molded in the exact material and surface texture intended for the final product.
4. It is not possible to manufacture a product to the *exact* dimensions placed on a product drawing because of changes in environmental conditions, wear of equipment and tools, and the variations in the ability and expertise of the people building the molds and operating the molding equipment.
5. All materials (except water) expand when they are heated, and shrink when they are cooled.
6. There are three basic groups of shrink factors for thermoplastic materials: *low*, *medium*, and *high*. Low shrink factors are considered those between 0.000 and 0.005 in./in. (0.000 and 0.127 mm/mm), medium are considered between 0.005 and 0.010 in./in. (0.127 and 0.254 mm/mm), and high are considered those between 0.010 and 0.015 in./in. (0.254 and 0.381 mm/mm).
7. Strength and support can be added to a boss without increasing wall thickness by placing ribs around the walls.
8. The flat-head screw design should not be considered because it may cause high stress conditions due to the wedging action of the screw head.
9. Proper thread diameter and size.
10. It is suggested that a metal insert be used if direct screw-to-plastic contact does not provide enough retention strength.

CHAPTER 12

1. The machine is a melt index unit and the test is ASTM #D-1238.
2. This test determines how far a plastic can be injected through a specific wall thickness of a product.

3. A crystalline nylon will flow easier than an amorphous polycarbonate.
4. This results in excessive and costly cycle times and more difficulty in controlling dimensions.
5. In its simplest terms, draft can be defined as a taper applied to side walls.
6. The molded product should have tapered walls to allow the part to eject easily from the mold.
7. 0.105 in. (2.667 mm).
8. If the pins are made too short they will cause a “pad” of excess plastic material to form on the part. If the pins are too long, the plastic will flow around them and cause a depression in the part.
9. The product drawing takes precedence over all other methods of conveying a product designer’s wishes to the manufacturing side of the industry.
10. Only notes that are specific to the particular product design.

CHAPTER 13

1. The main factor needed to determine how many cavities will be needed in a mold to produce a specific product is the annual volume requirement.
2. This awareness may lead the designer to select another plastic material, one which is more suited to the specific handling procedures.
3. The energy director.
4. Molecular structure.
5. Painting.
6. Flame treating.
7. It creates the possibility that mold release agents will be used to allow ejection of the part.
8. State on the drawing that mold release is not allowed in any manufacturing process associated with that product.
9. Bulk packaging.
10. Use of a plastic “sleeve” around the part, and packaging in individual cells.
11. Use a design that allows “nesting” of the parts in packaging.
12. UL (Underwriters Laboratories) and CSA (Canadian Standards Authority).

CHAPTER 14

1. The quandary at that point is: Who is responsible for a mistake and resultant defects?
2. If a customer has a good in-house molding facility but does not possess a mold design or build operation.
3. To present a design that meets the customer’s requirements.
4. Basic investigation, familiarity, and capabilities.
5. A messy plant indicates lack of concern for details and can result in late deliveries and poor-quality products.

6. Because these represent what you can expect for your own product.
7. Because older machinery is still quite capable of producing quality products.
8. That the vendor wishes to ensure accurate processing of materials, tool-making, traceability, and office procedures, including scheduling and quality control.
9. Because it helps ensure that all tooling will produce quality parts for the expected life span of the product.
10. The willingness of the vendor to participate and cooperate with the customer in establishing specific parameters, and controlling, reporting, and auditing of the methods and records pertaining to the customer's products.

CHAPTER 15

1. The primary reason for using a prototype is to achieve a comfort factor for form, fit, function, and appearance of a final product.
2. The disadvantages are cost and time.
3. By building a prototype.
4. Any of the following: There may be two or more components that must fit together to make the final product, or the final product is designed to fit within another product, or another product must fit within the plastic product being designed.
5. FEA stands for *finite element analysis*.
6. FEA programs are used to determine stress levels, analyze motion, check compressive and tensile factors, and just about any other functional attribute imaginable.
7. Proper appearance can make the difference between a successfully marketed product and a failure.
8. Texture samples cover a range of thousands of possible styles, from a faint satin finish to a heavy alligator hide.
9. In this process, plastic sheet and rod stock are machined and bonded together to create a facsimile of the desired product.
10. SLS stands for *selective laser sintering*.

CHAPTER 16

1. The injection mold is the heart of the injection molding process. It is where all of the forming action takes place.
2. The job of the sprue bushing is to seal tightly against the nozzle of the molding machine and to allow molten plastic to flow from the barrel into the mold.
3. Flash is a thin amount of molten plastic seepage. The two main causes are lack of proper clamp pressure and mold wear.

4. Hot runner systems eliminate waste generated by nonheated runners and allow faster cycles because the molding process does not have to wait for the runner to solidify to continue.
5. The three parts of the ejector pin are the body, the head, and the face.
6. By an ejector bar.
7. In the thickest section of the part.
8. A round cross-section is best.
9. When a mold is closed the air becomes trapped in the cavity openings and runner channels.
10. Vents can be placed in the mold.
11. To eliminate air and keep it from being pushed into the cavity where it would have to be vented anyway.

CHAPTER 17

1. Machine, mold, material, operator.
2. Machine = 60%, mold = 20%, material = 10%, operator = 10%.
3. 20% to 80%.
4. It is said to be “hygroscopic.”
5. It turns to steam.

List of Tables and Figures

Chapter 1

Table I-1 - Evolution of Injection Molding	2
Table I-2 - Suggested Melt Temperatures for Various Plastics	6
Table I-3 - Suggested Mold Temperatures for Various Plastics	10
Table I-4 - Melt Index Value's Impact on Plastics Properties	13
Table I-5 - Parameter Change versus Property Effect	24

Figure 1-1 - The injection molding process	4
Figure 1-2 - Categories of parameters	5
Figure 1-3 - Measuring plastic temperature	7
Figure 1-4a - The heating cylinder	7
Figure 1-4b - Nozzle heater zone	8
Figure 1-5 - Cylinder section showing screw action	9
Figure 1-6 - Controlling temperature of mold	11
Figure 1-7 - Melt index rheometer	12
Figure 1-8 - Mold-closing distances	16
Figure 1-9 - Injection and hold distances	17
Figure 1-10 - Cushion	18
Figure 1-11 - Ejection of finished part	19
Figure 1-12 - Horizontal molding machine	20
Figure 1-13 - Vertical molding machine	21
Figure 1-14 - Typical setup sheet	25
Figure 1-15 - Illustrating shrinkage rate	27
Figure 1-16 - How shrinkage affects dimensions	28
Figure 1-17 - Amorphous versus crystalline shrinkage differences	29
Figure 1-18 - Post-mold shrinkage fixture	31
Figure 1-19 - Defining stress	32

Chapter 2

Table II-1 - Amorphous versus Crystalline Properties	44
Table II-2 - Examples of Amorphous and Crystalline Materials	45
Table II-3 - Typical Elastomers	45
Figure 2-1 - Petroleum sources for common plastic materials	38
Figure 2-2 - An ethylene monomer	40
Figure 2-3 - Multiple ethylene monomers prior to polymerization	41
Figure 2-4 - POLYmerized ETHYLENE monomers (Polyethylene)	41

Figure 2-5 - Copolymer molecular structure (ABS)	42
Figure 2-6 - Amorphous molecular chains	43
Figure 2-7 - Crystalline molecular chains	44

Chapter 3

Table III-1 - Effects of 20% Glass Reinforcement on Polycarbonate	50
Table III-2 - Metal Fiber Characteristics	53
Table III-3 - Common Fillers and Their Uses	54

Chapter 4

Table IV-1 - Comparison of Crystalline, Amorphous, and Liquid Crystalline Materials	67
--	----

Figure 4-1 - The ordered symmetry of crystalline molecular structure	60
Figure 4-2 - Amorphous molecular structure is unstable and random	60
Figure 4-3 - Crystalline/amorphous molecular structure	61
Figure 4-4 - Differential Scanning Calorimeter graph of crystalline material	61
Figure 4-5 - DSC graph of amorphous material, highlighting the glass transition area	63
Figure 4-6 - Determining the glass transition point on the DSC graph	63
Figure 4-7 - Determining the percentage of crystallinity using DSC	64

Chapter 5

Table V-1 - Typical Volume Resistivity Values	86
Table V-2 - Typical Dielectric Constant Values	87
Table V-3 - Typical Dissipation Factor Values	88
Table V-4 - Chemical Resistance of Various Materials	90
Figure 5-1 - Shrinkage rate (mold shrinkage)	70
Figure 5-2 - Stress/strain curves	71
Figure 5-3 - Material exhibiting elasticity	72
Figure 5-4 - Material exhibiting plasticity	72
Figure 5-5 - Isotropic measurement	73
Figure 5-6 - Anisotropic measurement	74
Figure 5-7 - Specimen with stress applied	76
Figure 5-8 - Typical stress/strain (tensile) test	77
Figure 5-9 - Typical stress/strain curve plot	78
Figure 5-10 - Stress/strain curve elements	78

Figure 5-11 - Shear stress and shear strain defined	80
Figure 5-12 - Direct shear stress test	81
Figure 5-13 - Flexural strength fixture	82

Chapter 6

(Star diagrams)

Figure 6-1 - Acrylonitrile-butadiene-styrene (ABS)	108
Figure 6-2 - Acrylonitrile-butadiene-styrene/polycarbonate (ABS/PC)	109
Figure 6-3 - Acetal	110
Figure 6-4 - Acrylic	111
Figure 6-5 - Acrylic-styrene-acrylonitrile (ASA)	112
Figure 6-6 - Copolyester elastomer	113
Figure 6-7 - Liquid crystal polymer	114
Figure 6-8 - Nylon 6	115
Figure 6-9 - Nylon 12	116
Figure 6-10 - Nylon elastomers	117
Figure 6-11 - Polyarylate	118
Figure 6-12 - Polybutylene terephthalate (PBT)	119
Figure 6-13 - Polyethylene terephthalate (PET)	120
Figure 6-14 - Polycarbonate	121
Figure 6-15 - High-density polyethylene (HDPE)	122
Figure 6-16 - Polyketones	123
Figure 6-17 - Polypropylene (PP)	124
Figure 6-18 - High-impact polystyrene (HIPS)	125
Figure 6-19 - Sulfone polymers	126
Figure 6-20 - Polyphenylene oxide (PPO)	127
Figure 2-21 - Polyphenylene sulfide (PPS)	128
Figure 2-22 - Polyvinylchloride-rigid (PVC)	129
Figure 6-23 - Styrene-acrylonitrile (SAN)	130
Figure 6-24 - Styrene-maleic-anhydride (SMA)	131
Figure 6-25 - Thermoplastic polyolefin (TPO)	132
Figure 6-26 - Thermoplastic polyurethane (TPU)	133

Chapter 7

Table VII-1 - Average Machine Hour Rates	142
Figure 7-1 - Determining volume of parts and runner system	137

Chapter 8

Table VIII-1 - Ease of Welding (Amorphous Materials)	154
Table VIII-2 - Ease of Welding (Crystalline Materials)	155
Table VIII-3 - Adhesives for Bonding Common Plastics	162

Figure 8-1 - Ultrasonic welding machine	152
Figure 8-2 - Energy director formula	153
Figure 8-3 - Hot gas welding process	157
Figure 8-4 - Induction welding (electromagnetic) concept	158
Figure 8-5 - Spin welding	159
Figure 8-6 - Adhesive bond joint examples	161
Figure 8-7 - Drilling point details	163
Figure 8-8 - Design of turning points	165
Figure 8-9 - Spray painting	168
Figure 8-10 - Vacuum metallizing process	169
Figure 8-11 - Typical hot stamp machine	170
Figure 8-12 - Common roll-on decorating method	170
Figure 8-13 - Common peripheral marking method	171
Figure 8-14 - Typical vertical stamping method	171
Figure 8-15 - Typical pad printing machine	172
Figure 8-16 - Screen printing process	173
Figure 8-17 - Two-color molding concept	175
Figure 8-18 - In-mold overlay process	176

Chapter 9

Table IX-1 - Melt Index Value's Impact on Plastic Properties	188
Figure 9-1 - Dielectric strength testing	180
Figure 9-2 - Dielectric constant testing	181
Figure 9-3 - Volume resistivity testing	181
Figure 9-4 - Surface resistivity testing	181
Figure 9-5 - Arc resistance testing	182
Figure 9-6 - Shrinkage rate testing	182
Figure 9-7 - Determining the density of a plastic material	183
Figure 9-8 - Water absorption measurement	184
Figure 9-9a - TVI moisture content testing	186
Figure 9-9b - TVI patterns on test slide	187
Figure 9-10 - Melt flow index	187
Figure 9-11 - Tensile strength testing determines break and yield points in plastic materials	188
Figure 9-12 - Forces in compressive strength testing push toward a stationary sample until the material yields	188
Figure 9-13 - Flexural strength testing	189
Figure 9-14 - Creep testing	190
Figure 9-15 - Notched impact tests	191
Figure 9-16a - Crystalline melting temperature (T_m) graph, exhibiting its recognizable peak	192
Figure 9-16b - Amorphous glass transition (T_g) has no peak, but is recognizable by its S-shaped transition point ...	192

Figure 9-17 - Heat deflection temperature testing	192
Figure 9-18 - Vicat softening temperature testing	193
Figure 9-19 - Flammability testing	194
Figure 9-20 - Limiting oxygen index (LOI) testing	194
Figure 9-21 - Differential Scanning Calorimeter (DSC)	197
Figure 9-22a - Stress spike on DSC curve for a crystalline plastic	197
Figure 9-22b - Moisture spike on DSC curve	197
Figure 9-23 - Determining regrind percentage (crystalline material) ...	198
Figure 9-24 - Crystallinity curves	198
Figure 9-25 - Samples in furnace	199
Figure 9-26a - TGA chamber	200
Figure 9-26b - TGA curve	201

Chapter 10

Figure 10-1 - Product cost breakdown	208
Figure 10-2 - Product development chart	210
Figure 10-3 - Project management process	214
Figure 10-4 - Process management chart	215
Figure 10-5 - Product development time-line chart	217

Chapter 11

Table XI-1 - Recommended Tolerances for 6-in. (152-mm) Dimension	229
Table XI-2 - Typical Volume Resistivity Values	250
Figure 11-1 - Initial penholder concept	225
Figure 11-2 - Final penholder design	226
Figure 11-3 - Improper bolt assembly design	232
Figure 11-4 - Preferred bolt assembly design	233
Figure 11-5 - Preferred boss design concepts	234
Figure 11-6a - Assembling flat components with screws	235
Figure 11-6b - Controlling screw torque	236
Figure 11-7a - Proper part-to-boss location	237
Figure 11-7b - Self-tapping screw and boss design	238
Figure 11-8a - Examples of threaded inserts	239
Figure 11-8b - Typical insert designs for cold insertion	240
Figure 11-8c - Sonic-style insert installation	240
Figure 11-9 - Rivet/eyelet specifications	241
Figure 11-10 - Typical living-hinge design	242
Figure 11-11 - Pin-hinge design	243
Figure 11-12 - Common snap-fit (cantilever beam)	244
Figure 11-13a - Ultrasonic assembly machine	246
Figure 11-13b - Use of energy director	247

Figure 11-13c - Interference joint design	247
Figure 11-14 - Staking process	248
Figure 11-15 - Volume resistivity test	249
Figure 11-16 - Recycling symbol and chart identifying material types	251

Chapter 12

Table XII-1 - Typical Wall Thickness Ranges for Common Materials	257
Table XII-2 - Examples of Dimensional Change versus Draft Angles ...	267

Figure 12-1 - General design considerations	256
Figure 12-2a - Uniform wall thickness enables plastic molecules to flow without resistance	258
Figure 12-2b - Nonuniform wall thickness forces compression of plastic molecules, which results in stress	258
Figure 12-2c - Wall thickness transition	259
Figure 12-3a - Formation of a knit line	260
Figure 12-3b - Multiple-gate knit lines	261
Figure 12-4a - Stress concentration points resulting from sharp-corner design.	262
Figure 12-4b - Proper use of radii in sharp corners	262
Figure 12-5a - Sink mark caused by uniform wall thickness design	263
Figure 12-5b - Remedy for sink mark formation is a design calling for different wall thicknesses	263
Figure 12-6 - Draft angle defined	264
Figure 12-7a - With no draft angle, part ejection from the mold is very difficult	265
Figure 12-7b - Just 1° of draft permits sufficient air to enter the mold to enable easy ejection of the part	266
Figure 12-8 - Designing with no draft	268
Figure 12-9 - Special mold design for no-draft parts	269
Figure 12-10 - Ejector pin witness marks	270
Figure 12-11 - Gate witness marks	271
Figure 12-12 - Radii versus sharp corners. Uniform wall thickness throughout the corners ensures uniform shrinkage, thus eliminating stress	279
Figure 12-13 - Plastic product design checklist	280-281

Chapter 14

Figure 14-1 - Plastic molding vendor qualification form	300-303
---	---------

Chapter 15

Table XV-1 - Rapid Prototype Equipment Manufacturers 312

Chapter 16

Figure 16-1 - "A" and "B" plates of a two-cavity mold 316

Figure 16-2 - The "A" half of the mold 317

Figure 16-3 - Ejector pins such as these facilitate part ejection 318

Figure 16-4 - Ejector plate device showing ejector pin locking
concept 319

Figure 16-5 - Ejector plates actuated by knockout rod 320

Figure 16-6 - Locating the gate at the thickest section of the part
reduces potential for stress 321

Figure 16-7 - Comparing round runner cross-section with
trapezoid 322

Figure 16-8 - Air trapped by incoming flow 323

Figure 16-9 - Venting of parting line 324

Figure 16-10 - Venting of the runner 325

Chapter 17

Figure 17-1 - Distribution of defect causes 333

Appendix A

Material Tradenames and Descriptions*

Tradename	Material	Manufacturer
Aclon	fluoropolymer	Allied Signal
Acrylite	acrylic	Caro
Adell	TP resins	Adell
Adpro	polypropylene	Novacor
Affinity	polyolefin plastomers	Dow Plastics
Aim	various plastics	Dow Plastics
Akulon	nylon	A. Schulman
Alathon	polyethylene	DuPont
Albis	nylon	Albis
Alcryn	TP elastomer	DuPont
Algoflon	fluoropolymer	Ausimont
Amilan	nylon	Toray
Amoco	TP resins	Amoco
Amodel	polyphthalamide	Amoco
Apec	polycarbonate	Bayer
Aquathene	polyethylene	Quantum
Arpak	expandable PP bead	ARCO
Arpro	expandable PP bead	ARCO
Astryn	filled PP	Montell
Attane	linear LDPE	Dow
Aurum	polyimide	Mitsui Toatsu
Azdel	TP resins	GE Plastics

*Portions of this information provided by Christopher Bryce, Texas Plastic Technologies; and Steve Spanoudis, Plastics Technology Center of Lexmark. Used with their permission.

Tradename	Material	Manufacturer
Bapolene	polyethylene	Bamberger
Barex	acrylonitrile coploymer	BP Chemicals
Bayblend	polycarbonate/ABS	Bayer
Baydur	st. foam PUR (RIM)	Bayer
Baylon	6/6 nylon	Bay Resins
Beetle	thermosets	Cytec Industries
Benvic	PVC	Solvay
Bexloy	ionomer	DuPont
Boltaron	FR PP	GenCorp
Cabot	TP resins	Cabot
Cadon	SMA coploymer	Monsanto
Calibre	polycarbonate	Dow
Capron	nylon	Allied Signal
Carilon	aliphatic PK	Shell
Cefor	polypropylene	Shell
Celanese	6 and 6/6 nylons	Hoechst-Celanese
Celanex	PBT	Hoechst-Celanese
Celcon	acetal polymer	Hoechst-Celanese
Cevian	ABS	Hoechst-Celanese
Chemigum	TP elastomer	Goodyear
Claradex	ABS	Shin-A
Comtuf	impact resistant resins	ComAlloy
Corton	mineral-filled resins	PolyPacific
Crastin	PMP	Phillips 66
Cycogel	ABS	Nova
Cycolac	ABS	General Electric
Cycolin	ABS/PBT	General Electric
Cycoloy	PC/ABS	General Electric
Cycovin	ABS/PVC	General Electric
Cyrex	acrylic/polycarbonate	Cyro
Cyrolite	acrylic polymers	Cyro
Delrin	acetal	DuPont
Desmopan	polyurethane	Bayer
Dowlex	LDPE	Dow
Drexflex	TP elastomer	D & S Plastics
Duraflex	polybutylene	Shell
Dural	vinyl, PVC	Alpha Chemical
Durethan	nylon 6	Bayer
Dylark	SMA copolymer	ARCO
Dylite	expandable PS bead	ARCO
Eastman	TP resins	Eastman Chemical
Edistir	polystyrene	Enichem

Tradename	Material	Manufacturer
Ektar	PET, PBT, PCT	Eastman Chemical
Ektar FB	TP elastomer	Eastman Chemical
Elastalloy	TP elastomer	GLS Corp.
Elastolan	polyurethane TPE	BASF
Elvamide	nylon copolymer	DuPont
Eltex	HDPE	Solvay
Eltex P	PP	Solvay
Elvax	EVA copolymer	DuPont
Emac	EMA copolymer	Chevron
Empee	PE, PP	Monmouth
Enathene	ethylene butyl acrylate	Quantum
Engage	TP elastomer	Dow
Eref	PA/PP alloy	Solvay
Escor	acid terpolymer	Exxon
Escorene	PP	Exxon
Esthane	polyurethane TPE	BF Goodrich
Exxtral	TP elastomer	Exxon
Ferrene	PE	Ferro
Ferroflo	PS	Ferro
Ferropak	PP/PE alloy	Ferro
Fiberloc	fiber-reinforced PVC	Geon
Fina	polyolefin	Fina Oil
Finaclear	PS	Fina Oil
Finaprene	TP elastomer	Fina Oil
Flexomer	ULDPE	Union Carbide
Fluorocomp	reinforced fluoropolymer	LNP
Formion	ionomer	A. Schulman
Fortiflex	PE	Solvay
Fortilene	PP	Solvay
Fortron	PPS	Hoechst-Celanese
Geon	PVC	Geon
Grilamid	nylon	EMS Grilon
Halar	fluoropolymer	Ausimont
Halon	fluoropolymer	Ausimont
Hanalac	ABS	Miwon
Hercuprene	TP elastomer	J-Von
Hifax	TP elastomer	Montell
HiGlass	glass-filled PP	Himont
HiVal	HDPE	General Polymers
Hostaflon	PTFE	Hoechst-Celanese
Hostalen	UHMWPE	Hoechst-Celanese
Hostalloy	PE alloy	Hoechst-Celanese

Tradename	Material	Manufacturer
Hyflon	fluoropolymer	DuPont
Hylar	PVDF	Ausimont
Hyterl	TP elastomer	DuPont
Impet	polyester	Hoechst-Celanese
Iotek	ionomer	Exxon
Isoplast	TPU	Dow
Iupiace	PPO/PPE	Mitsubishi
Iupilon	polycarbonate	Mitsubishi
Iupital	acetal	Mitsubishi
Ixan	PVDF	Solvay
Ixef	polyarylamide	Solvay
J-Plast	TP elastomer	J-Von
Kadel	PAEK	Amoco
Kamax	acrylic copolymer	AtoHaas
Koblend	PC/ABS	EniChem
Kodapak	PET polyester	Eastman Chemical
Kodar	PETG polyester	Eastman Chemical
Kohinor	vinyl	Rimtec
Kraton	styrenic TPE	Shell
K-Resin	styrene/butadiene	Phillips 66
Kynar	PVDF	Elf Atochem
Ladene	PS	SABIC
Lexan	polycarbonate	General Electric
Lomod	TP elastomer	General Electric
Luran	SAN, ASA	BASF
Lusix	nylon 6/6	Westover
Lustran	ABS, SAN copolymer	Monsanto
Magnum	ABS	Dow
Makroblend	polycarbonate blend	Bayer
Makrolon	PC, PC blends	Bayer
Malecca	styrenic copolymer	Denki Kagaku
Maranyl	nylon	ICI Americas
Marlex	PE, PP	Phillips 66
Milastomer	TP elastomer	Mitsui
Mindel	PSO alloy	Amoco
Minlon	mineral-filled nylon	DuPont
Morthane	TP elastomer	Morton
Multibase	ABS	Multibase
Multi-Flam	PP	Multibase
Multi-Flex	TP elastomer	Multibase

Tradename	Material	Manufacturer
Multi-Hips	high-impact PS	Multibase
Multi-Pro	PP	Multibase
Multi-San	SAN copolymer	Multibase
Naxel	recycled PC	MRC Polymers
Nortuff	PP	Quantum
Noryl	PPO/PPE	General Electric
Novablend	PVC	Novacor
Novalast	TP elastomer	Nova Polymers
Novalene	TP elastomer	Nova Polymers
Novamid	nylon	Mitsubishi
Novapol	PE	Novacor
Novon	starch-based polymer	Novon
Nucrel	EMAA copolymer	DuPont
Nydur (Durethan)	nylon 6	Bayer
Nylamid	nylon	Polymer Service
Nylatron	glass-reinforced nylon	DSM
Olehard	filled PP	Chiso America
Optema	EMA copolymer	Exxon
Optix	acrylic	Plaskolite
Oxy	vinyl	Occidental
Oxyblend	vinyl	Occidental
Oxyclear	PVC	Occidental
Panlite	polycarbonate	Teijin Chemical
Paxon	TP resins	Paxon
Pebax	TP elastomer	Elf Atochem
Pellethane	PUR TP elastomer	Dow
Perspex	acrylic	ICI Acrylics
Petlon	polyester	Albis
Petra	PET	Allied Signal
Petrothene	PE, PP, EVA	Quantum
Pibiter	PBT	EniChem
Plexiglas	acrylic	AtoHaas (Rohm & Haas)
Pliovic	vinyl	Goodyear
Pocan	PBT	Albis
Polifil	reinforced polyolefins	Polifil
Polyfabs	ABS	A. Schulman
Polyflon	fluoropolymer	Daikin
Polyfort	PE, PP	A. Schulman
Polyman	ABS alloy	A. Schulman
Polypur	alloyed TPE	A. Schulman
Polytrope	TP elastomer	A. Schulman
Prevail	ABS/polyurethane	Dow

Tradename	Material	Manufacturer
Prevex	PPO/PPE	General Electric
Primef	PPS	Solvay
Prism	polyurethane (RIM)	Bayer
Polyvin	PVC	A. Schulman
Primacor	EAA copolymer	Dow
Pro-Fax	polyolefins	Montell
Propak	PP	PolyPacific
Pulse	PC/ABS	Dow
Radel	polyether sulfone	Amoco
Radiflam	nylon FR	Polymers International
Radilon	nylon 6	Polymers International
Radipol	nylon 6/6	Polymers International
Reny	nylon 6/6	Mitsubishi
Replay	PS	Huntsman
Repreat	ethylene copolymer	Discas
Rexene	TP resins	Rexene
Rexflex	PP	Rexene
Rimplast	TP elastomer	Huls
Rimtec	vinyl	Rimtec
Riteflex	TP elastomer	Hoechst-Celanese
Rynite	PET	DuPont
Ryton	PPS	Phillips 66
Saran	vinylidene chloride	Dow
Sarlink	TP elastomer	DSM
Schulaflex	flexible elastomers	A. Schulman
Schulamid	nylon	A. Schulman
Schulink	cross-linkable HDPE	A. Schulman
Sclair	PE	DuPont Canada
Shell	polyolefins	Shell
Sinkral	ABS	EniChem
Sinvet	polycarbonate	EniChem
Soarnol	EVA copolymer	Nichimen
Solef	PVDF	Solvay
Solvic	PVC	Solvay
Spectar	polyester copolymer	Eastman Chemical
Stanuloy	recycled PC/PET blend	MRC Polymers
Stapron	SMA copolymer	DSM
Statcon	static dissipative resin	LNP
Stereon	styrene/butadiene	Firestone
Styron	PS	Dow
Styropor	PS	BASF
Sunprene	PVC elastomer	A. Schulman
Supec	PPS	General Electric

Tradename	Material	Manufacturer
Suprel	ABS/PVC	Vista
Surlyn	ionomer	DuPont
Teflon	PTFE	DuPont
Tefzel	PE/TFE	DuPont
Telcar	TP elastomer	Teknor Apex
TempRite	CPVC	BF Goodrich
Tenac	acetal	Asahi
Tenite	CAB, polyolefins	Eastman Chemical
Texin	PUR	Bayer
Toplex	PC/ABS	Multibase
Torlon	polyamide-imide	Amoco
Toyolac	ABS, PC/ABS	Toray
TPX	PMP	Mitsui
Triax	PC/ABS, ANS/Nylon	Monsanto
Tyrl	SAN	Dow
Udel	PSO	Amoco
Ultem	PEI	General Electric
Ultradur	PBT	BASF
Ultraform	acetal	BASF
Ultramid	nylon	BASF
Ultrapek	PAEK	BASF
Ultrason-E	PES	BASF
Ultrathene	EVA copolymer	Quantum
Unichem	PVC	Colorite
Unival	PE	Union Carbide
Valox	PBT	General Electric
Valtra	PS	Montell
Vandar	polyester alloy	Hoechst-Celanese
Vespel	polyimide	DuPont
Vestamid	nylon	Huls
Victrex	PEEK	Victrex
Vista	vinyl	Vista
Vistel	PVC	Vista
Vydyne	nylon	Monsanto
Vyram	TP elastomer	Advanced Elastomer
Wellamid	nylon	Wellman
Xenoy	PC/PBT	General Electric
XT-Polymer	acrylic	Cyro
Xydar	liquid crystal polymer	Amoco

Tradename	Material	Manufacturer
Zanite	liquid crystal polymer	DuPont
Zeonex	PMP	Nippon Zeon
Zylar	acrylic	Novacor
Zytel	nylon	DuPont

Supplier Locations

Adell	Baltimore, MD
Advanced Elastomer Systems	St. Louis, MO
Albis	Rosenburg, TX
Allied Engineered Plastics	Morristown, NJ
Allied Signal	Morristown, NJ
Alpha Chemicals & Plastics	Pineville, NC
Amoco Chemical Co.	Chicago, IL
ARCO Chemical	Newton Square, PA
Asahi America	Malden, MA
AtoHaas North America	Philadelphia, PA
Ausimont	Morristown, NJ
BASF Corp.	Parsippany, NJ
Bay Resins	Millington, MD
Bayer	Pittsburgh, PA
BP Chemicals	Hackettstown, NJ
Cabot	Stratford, CT
Chevron Chemical Co.	Houston, TX
Colorite Polymers	Burlington, NJ
ComAlloy	Nashville, TN
Cyro Industries	Orange, CT
Cytec Industries (formerly Cyanimid)	West Paterson, NJ
D & S Plastics	Auburn Hills, MI
DSM Engineering Plastics	Evansville, IN
Daikin America	Orangeburg, NY
Denki Kagaku Kagyo	New York, NY
Dow Chemical Co.	Midland, MI
DSM Engineering Plastics	Evansville, IN
DuPont de Nemours & Co.	Wilmington DE
EMS-American Grilon	Sumter, SC
Eastman Chemical Co.	Kingsport, TN
Elf Atochem North America	Philadelphia, PA
Exxon Chemical Co.	Houston, TX

Fina Oil	Dallas, TX
Firestone Synthetic Rubber & Latex Co.	Akron, OH
GLS Corp.	Cary, IL
GenCorp	Newcomerstown, OH
General Electric Co.	Pittsfield, MA
General Polymers	Dublin, OH
Geon Co.	Cleveland, OH
B.F. Goodrich	Cleveland, OH
Goodyear	Akron, OH
Himont, Inc.	Wilmington, DE
Hoechst-Celanese	Chatham, NJ
Huls America	Farmington Hills, MI
Huntsman	Houston, TX
ICI Chemicals & Polymers	Wilmington, DE
J-Von, L.P.	Leominster, MA
LNP Engineering Plastics	Exton, PA
MRC Polymers	Chicago, IL
Miles, Inc.	Pittsburgh, PA
Mitsubishi Engineering Plastics	White Plains, NY
Mitsui Toatsu	Tokyo, Japan
Monsanto	St. Louis, MO
Montell	Wilmington, DE
Morton International	Seabrook, NH
Multibase, Inc.	Coply, OH
Nova	Eatontown, NJ
Novacor Chemicals, Inc.	Leominster, MA
Novon International	Tonawanda, NY
Nylon Corp. of America	Manchester, NH
Occidental Chemical Corp.	Dallas, TX
Oxychem	Houston, TX
Paxon Polymer Co.	Baton Rouge, LA
Phillips 66 Co.	Bartlesville, OK
Plaskolite, Inc.	Columbus, OH
Polyfil	Woonsocket, RI
Polymer Service	Clinton Township, MI
Polymerland, Inc.	Parkersburg, WV
Prime Alliance	Minneapolis, MN
Quantum Chemical	Cincinnati, OH

Rexene
Rimtec Corp.
RTP Co.

Dallas, TX
Burlington, NJ
Winona, MN

A. Schulman Inc.
Shell Chemical Co.
Shiñ-A
Solvay

Akron, OH
Houston, TX
Kyunggido, Korea
Houston, TX

Teknor Apex
Thermofil
Toray

Pawtucket, RI
Brighton, MI
San Mateo, CA

Union Carbide Corp.

Danbury, CT

Victrex USA
Vista Chemical Co.

West Chester, PA
Houston, TX

Wellman, Inc.

Johnsonville, SC

Appendix B

Common Material Acronyms and Their Meanings*

A - Allyl	CMC - carboxymethyl cellulose
ABA - acrylonitrile butadiene acrylate	CN - cellulose nitrate (celluloid)
ABS - acrylonitrile butadiene styrene	COP - copolyester thermoplastic elastomer
ACPES - acrylonitrile chlorinated polyethylene styrene	CP - cellulose propionate
AEPDM - acrylonitrile ethylene propylene diene styrene	CPE - chlorinated polyethylene
AES - acrylonitrile ethylene styrene	CPVC - chlorinated polyvinyl chloride
AMA - acrylate maleic anhydride	CS - casein
AMAB - acrylonitrile-methylacrylate acrylonitrile-butadiene	CTA - cellulose triacetate
AMMA - acrylonitrile methyl methacrylate	CTFE - chlorotrifluoroethylene
AN - acrylonitrile	DAIP - diallyl isophthalate
APO - amorphous polyolefin	DAP - diallyl phthalate
ARP - aromatic polyester	EA - ethylene acrylic acid copolymer
ASA - acrylic styrene acrylonitrile	EBA - ethylene butyl acrylate
BMC - Bulk Molding Compound	EC - ethyl cellulose
BMI - bismaleimide	ECTFE - ethylene chlorotrifluoroethylene
CA - cellulose acetate	EEA - ethylene ethyl acetate
CAB - cellulose acetate butyrate	EMAC - ethylene methacrylate acid copolymer
CAP - cellulose acetate propionate	EP - epoxy
CF - cresol formaldehyde	EPDM - ethylene propylene diene monomer
CFC - chlorofluorocarbon	EPM - ethylene propylene monomer
	EPR - ethylene propylene rubber
	EPS - expandable polystyrene

*This information provided by the ASTM Standard Technology for Abbreviated Terms (D 1600-92); Christopher Bryce, Texas Plastic Technologies; and Steve Stanopoulos, Plastics Technology Center of Lexmark. Reprinted with their permission.

ETFE - ethylene tetrafluoroethylene	PAN - polyacrylonitrile
EVA - ethylene vinyl acetate	PARA - polyaryl amide
EVAL - ethylene vinyl alcohol	PAS - polyarylsulfone
E/VCA - ethylene/vinyl acetate copolymer	PB - polybutylene
EVOH - ethylene vinyl alcohol (alternate spelling)	PBA - polybutyl acrylate
	PBAN - polybutadiene acrylonitrile
FEP - fluorinated ethylene propylene	PBD - polybutadiene
FF - furanformaldehyde	PBS - polybutadiene styrene
FRP - fiber-reinforced plastic	PBT - polybutadiene terephthalate
	PC - polycarbonate
HDPE - high-density polyethylene	PC/ABS - polycarbonate/acrylonitrile butadiene styrene (blend)
HIPS - high-impact polystyrene	PCT - polycyclohexyl terephthalate
HMWHDPE - high-molecular weight, high-density, polyethylene	PCTFE - polymonochlorotrifluoroethylene
	PCT-G - glycol-modified polycyclohexyl terephthalate
I - ionomer	PE - polyethylene
IPN - interpenetrating polymer network	PEBA - polyether block amide
	PEEK - polyetheretherketone
LCP - liquid crystal polymer	PEI - polyetherimide
LDPE - low-density polyethylene	PEK - polyetherketone
LLDPE - linear, low-density polyethylene	PEN - polyethylene naphthalene
LMDPE - linear, medium-density polyethylene	PEO - polyethylene oxide
LPE - linear polyethylene	PES - polyethersulfone
	PET - polyethylene terephthalate
MA - maleic anhydride	PETG - glycol-modified polyethylene terephthalate
MBS - methacrylate butadiene styrene	PEUR - polyether urethane
MC - methyl cellulose	PF - phenol formaldehyde
MDPE - medium-density polyethylene	PFA - perfluoroalkoxy alkane
MF - melamine formaldehyde	PFF - phenol furfural
MMA - methyl methacrylate	PI - polyimide
MPF - melamine phenol formaldehyde	PI - polyisoprene
	PIB - polyisobutylene
OPP - oriented polypropylene	PIR - polyisocyanurate
OPS - oriented polystyrene	PISU - polyimide sulfone
OSA - olefin modified styrene acrylonitrile	PMAN - polymethacrylonitrile
	PMCA - polymethyl chloroacrylate
P - phenolic	PMMA - polymethylmethacrylate (acrylic)
PA - polyamide (nylon)	PMP - polymethylpentene
PA - polyaniline	PMS - poly-methyl styrene
PAA - poly-acrylic acid	POM - polyoxymethylene (polyacetal)
PADC - polyallyl diglycol carbonate	PP - polypropylene
PAE - polyaryl ether	PPA - polyphthalamide
PAEK - polyaryletherketone	PPE - polyphenylene ether
PAI - polyamide-imide	PPI - polymeric polyisocyanate
PAK - polyester alkyd	PPO - polyphenylene oxide

PPOX - polypropylene oxide	TES - thermoplastic elastomer, styrenic
PPS - polyphenylene sulfide	TFE - tetrafluoroethylene
PPSU - polyphenylene sulfone	TMC - thick molding compound
PPT - polypropylene terephthalate	TPE - thermoplastic elastomer
PS - polystyrene	TPES - thermoplastic polyester
PS-b-PI - polystyrene/polyisoprene block copolymer	TPO - thermoplastic polyolefinic elastomer
PSO, PSU - polysulfone	TPU(R) - thermoplastic polyurethane
PTFE - polytetrafluoroethylene	TPX - polymethylpentene
PTMT - polytetramethylene terephthalate	TSUR - thermoset polyurethane
PU, PUR - polyurethane	TVO - thermoplastic vulcanites
PVA - polyvinyl alcohol	UF - urea formaldehyde
PVAC - polyvinyl acetate	UHMWPE - ultra-high-molecular weight polyethylene
PVB - polyvinyl butyryl	ULDPE - ultra-low-density polyethylene
PVC - polyvinyl chloride	UP - unsaturated polyester
PVCA - polyvinyl chloride acetate	
PVDC - polyvinylidene chloride	VA - vinyl acetate
PVDF - polyvinylidene fluoride	VAE - vinyl acetate ethylene
PVF - polyvinyl fluoride	VCE - vinyl chloride ethylene
PVK - polyvinyl carbazole	VCEMA - vinyl chloride ethylene methyl acrylate
PVOH - polyvinyl alcohol	VCEV - vinyl chloride ethylene vinyl-acetate
PVP - polyvinyl pyrrolidone	VCM - vinyl chloride monomer
S - styrene	VCMA - vinyl chloride methyl acrylate
SAN - styrene acrylonitrile	VCMAA - vinyl chloride methyl methacrylate
SB - styrene butadiene	VCOA - vinyl chloride octyl acrylate
SBR - styrene butadiene rubber	VCVAC - vinyl chloride-vinyl acetate chloride
SF - structural foam	VCVDC - vinyl chloride-vinylidene chloride
SHIPS - super-high-impact styrene	VDC - vinylidene chloride
SI - silicone	
SMA - styrene maleic anhydride	XLPE - cross-linked polyethylene
SMC - sheet molding compound	XPS - expandable polystyrene
SMS - styrene-methyl styrene	
SP - saturated polyester	
SRP - styrene rubber plastics	
TEEE - thermoplastic elastomer, ether-ester	
TEO - thermoplastic elastomer, olefinic	

Appendix C

Material Rankings for Various Properties

The following listing shows the relative ranking (lowest to highest) of common plastic materials for various property values. These are indicators only. Glass-reinforced materials are not listed and rankings for them may be different. Contact your material supplier representative for detailed information on a specific material or grade.

Compressive Strength

Fluorocarbon (PTFE) 1	Polyethylene 2	Cellulose propionate 3
Polyethylene (LD) 4	Cellulose acetate 5	Cellulose butyrate 6
Polypropylene 7	Nylon 6 8	Nylon 6/6 9
Acetal 10	ABS 11	PVC 12
Polyester (TP) 13	SAN 14	Polycarbonate 15
Polystyrene 16	Acrylic 17	Polysulfone 18
Polypheylene sulfide . 19	Polyphenylene oxide ... 20	

Tensile Strength

Fluorocarbon 1	PVC 2	Polyethylene 3
Polyurethane 4	Cellulose butyrate 5	Cellulose propionate 6
Cellulose acetate 7	Polyester 8	Polypropylene 9
Polystyrene 10	ABS 11	Acetal 12
Acrylic 13	PPO 14	Polyether sulfone 15
Polycarbonate 16	Nylon 6 17	PPS 18
Polysulfone 19	SAN 20	

Flexural Strength

Polyurethane 1	Cellulose propionate 2	Cellulose butyrate 3
PVC 4	Cellulose acetate 5	Polyethylene (LD) 6
Polypropylene 7	Polystyrene 8	Nylon 6 9
ABS 10	Acetal 11	PPO 12
Nylon 6/6 13	Polyester 14	Acrylic 15
Polycarbonate 16	Polysulfone 17	Polyether sulfone 18
PPS 19	SAN 20	

Tensile Modulus

Fluorocarbon	1	Polyurethane	2	Cellulose butyrate	3
Cellulose propionate	4	Polyethylene	5	Cellulose acetate	6
PVC	7	Polyester	8	Polypropylene	9
Polyethylene (HD)	10	Polyether sulfone	11	Nylon 6/6	12
Polysulfone	13	Nylon 6	14	Acetal	15
Acrylic	16	Polystyrene	17	ABS	18
PPO	19	PPS	20		

Flexural Modulus

Polyurethane	1	Polyethylene (LD)	2	Cellulose butyrate	3
Cellulose propionate	4	Polyethylene (HD)	5	Cellulose acetate	6
Polypropylene	7	PVC	8	Polyester	9
Polyether sulfone	10	Acetal	11	Polysulfone	12
PPO	13	Nylon 6	14	ABS	15
Polycarbonate	16	Polystyrene	17	Acrylic	18
PPS	19	SAN	20		

Heat Deflection Temperature

Polysulfone	1	Polyether sulfone	2	Polycarbonate	3
PPS	4	Acetal	5	ABS	6
Polystyrene	7	Nylon 6/6	8	PVC	9
Acrylic	10	SAN	11	Nylon 6	12
Cellulose propionate	13	Cellulose butyrate	14	Polypropylene	15
Cellulose acetate	16	Fluorocarbon	17	Polyethylene (LD)	18
Polyester	19	Polyethylene (HD)	20		

INDEX

Index Terms

Links

A				
acetal	2	6	10	45
	86	88	95	110
	166	168	229	250
	252	257		
acetate	96	229		
acid etch	167			
acrylate(s)	45			
butadiene	104			
acrylic(s)	6	10	45	86
	88	96	111	156
	168	229	250	257
rubber	104			
-styrene-acrylonitrile (ASA)	96	112		
acrylonitrile-butadiene-styrene (ABS)	2	6	10	41
	42	44	45	54
	55	86	96	106
	108	154	156	167
	168	175	229	241
	248	250	257	287
/polycarbonate	109			
adhesive bond(ing)	159	161		
alkyd	168			

Index Terms

Links

alloys	42	47		
American Society for Testing and Materials (ASTM)	103	179		
standard				
#D-1238	12			
#D-618	180			
#D-149	180			
#D-150	180			
#D-257	180			
#D-495	181			
#D-955	182			
amorphous	28	43	83	198
materials	4	45	59	62
	63	65	69	84
	151			
molecular				
chains	43	44		
structure	59	60		
properties	44			
shrinkage	29			
“angle pins,”	268			
anisotropy	73			
antioxidants	55			
antistatic agents	56			
applied finishes	167			
aramid	51			
arc resistance	88			
assembly	233			
methods	231			

This page has been reformatted by Knovel to provide easier navigation.

Index Terms

Links

assembly (<i>Cont.</i>)				
operations	151			
automated shape cutting	165			
B				
Ballistic Particle				
Manufacturing (BPM)	311			
barrel	7			
capacity	141	333		
rear temperature	24			
temperature	29	336	340	342
zone	24			
black specks	333			
blends	42	47		
blisters	334			
blowing agents	56			
blush	334			
boron	51			
bowing	262	334		
Branson				
Ultrasonics Corp.	153			
brittleness	72	335		
bubbles (voids)	335			
burn marks	335			
butadiene rubbers	105			
butyl(s)	45			
rubber	104			
butyrate	96	97		

Index Terms

Links

C

Canadian Standards Association (CSA)	277	288		
carbon-graphite	52			
carboxylic	105			
casting	310			
cavity(ies)				
image(s)	13	15	29	315
	316	320	323	334
mold	12	176		
celluloid	1	2		
cellulose	96			
acetate	6	10		
butyrate	6	10	45	
propionate	6	10	45	
cellulosics	168			
ceramic(s)	52			
chemical(s)	89			
reaction	89			
resistance	13	67	90	
solvation	89			
chlorosulfonated polyethylene	45			
clamp				
force	140			
mechanism	4			
pressure	14	140		
tonnage	140	141		
unit	4	13		
clear spots	336			

Index Terms

Links

cloudy appearance	336			
coefficient of linear thermal expansion (CLTE)	84			
color	308			
colorability	97			
colorant(s)	54	156	174	
Comparative Tracking Index (CPI)	88			
concurrent engineering	216			
contamination	334	337	338	
contour	308			
controlled pressure molding	176			
cooling				
post-mold	30	31		
copolymer(s)	41	42		
butadiene-acrylonitrile	104			
chlorotrifluoroethylene	105			
ethylene-propylene	104			
hexafluoropropylene	105			
isobutylene-isoprene	104			
phenylene ether	106			
vinylidene fluoride	105			
copolyester elastomers	113			
corona discharge	167			
cracking	33	337	338	
crazing	33	338		
creep	82	101	189	
resistance	13	67	83	92
	187			
testing	189			

This page has been reformatted by Knovel to provide easier navigation.

Index Terms

Links

cross-linking	46			
crystalline	8	28	43	47
	197			
materials	5	45	46	59
	62	64	69	84
	98	151	198	246
	250			
molecular				
chains	44	64		
structure	59	60	66	
properties	44			
shrinkage	29			
crystalline/amorphous				
molecular structure	60	61		
cushion (pad)	18			
cycle time(s)	30	142	143	146
	159	318	326	
gate-to-gate	21			
cylinder	3			
heating	4	5		

D

decorative procedures	166
defects	331
delamination	338
density	69
design considerations	276
destaticizers (<i>see antistatic agents</i>)	

Index Terms

Links

dielectric				
constant (permittivity)	87			
properties	249			
strength	87			
Differential Scanning Calorimeter				
(DSC)	61	67	83	190
	196			
discoloration	338			
dissipation factor	87			
values	88			
distance	4	15		
ejection	20			
holding	17			
initial injection	17			
mold-closed	16			
mold-open	18	19		
draft	264	287		
allowance and tolerances	299			
angle(s)	231	264	337	
limitations	275			
drilling				
thermoplastics	160			
thermosets	164			
ductility	72			

E

ejector				
bar	319	320		

Index Terms

Links

ejector (<i>Cont.</i>)				
marks	269	270	272	
pin(s)	19	318	319	
plates	319	320		
rod	320			
ejector retainer plate	319			
elastic limit	79			
elasticity	71	72		
elastomeric(s)	159			
adhesives	160			
elastomers	44	45	103	
electrical discharge machine (EDM)	315			
electrical properties	85			
electromagnetic interference (EMI)	53			
elongation	50	67		
energy director(s)	152	153	245	247
environmental				
concerns	88			
issues	251			
epichlorohydrin rubbers	106			
epoxy(ies)	159	168		
equipment	297			
ethyl cellulose	96	97		
ethylene	40			
rubber	104			
terpolymer rubber	104			
exposure factors	88			
monomer	41			
eyelets	241			

This page has been reformatted by Knovel to provide easier navigation.

Index Terms

Links

F

fabrication	309			
failure analysis (FA)	179	195		
fiberglass	49	50	200	
fillers	49	53	54	156
finite element analysis (FEA)	308			
flame				
retardants	55	56	156	
treatment	166	287		
flammability	85	249		
testing	193	194		
flash	26	318	339	
flow				
index rating	12			
lines	339			
rate(s)	141	154	155	339
flowability	67			
fluorocarbon(s)	45	160	168	
fluoroelastomers	105			
fluoroplastics	97			
fluorosilicones	45			
foaming agent(s) (<i>see blowing agents</i>)				
“freeze off,”	14			
friction welding (<i>see spin welding</i>)	159			

G

gate	15	28	334	341
design	340			

This page has been reformatted by Knovel to provide easier navigation.

Index Terms

Links

gate (*Cont.*)

location 320

location restrictions 269

sizing 336

witness marks 270

glass content 199

glass transition (T_g)

area 63

point 62 63 84

range 153

temperatrure 62

gloss 13 339

H

hardness 13

heat deflection temperature (HDT) 50 84 191 248

heat exchanger 11

heat resistance 66

heat stabilizer(s) 55

heater band(s) 2 3 5 7

heating cylinder 7 8 332 334

Hooke's Law 75 79

hopper 4 8 9 51

337 340

horizontal molding 20

hot gas welding 157

hot stamp(ing) 169

machine 170

Index Terms

Links

hygroscopic	154	335	342
materials	72	250	
Hypalon™	104		
I			
induction (electromagnetic) bonding	157		
injection			
cylinder	2		
molding			
costs	135		
process	4	15	17
pressure	140	141	
unit	4	14	
in-mold			
overlay(s)	175		
process	176		
in-process finishes	173		
International Organization for			
Standardization (ISO)	179		
standard			
9000	299		
#IEC 243-1	180		
#IEC 250	180		
#IEC 93	180		
#IEC 294	182		
ionomers	97		
isotropy	73		

Index Terms

Links

J

Jetting	340	
John Wesley Hyatt	1	2
Just-in-Time (JIT)	209	

K

knit lines(s)	258	340
knockout rod	19	319

L

labor		
charges	136	
costs	139	
Laminated Object Manufacturing (LOM)	311	
laser cutters	166	
limiting oxygen index (LOI)	194	
liquid crystalline		
materials (plastics)	67	83
living hinge(s)	241	242
low-pressure molding	176	
lubricants	56	
lubricity	73	

M

machine		
costs	139	

This page has been reformatted by Knovel to provide easier navigation.

Index Terms

Links

machine (<i>Cont.</i>)				
fixed	140			
variable	139			
man hour rate (MHR)	142	147		
rate	136			
machining operations	160			
maintenance costs	145			
material(s)	297			
applications	95			
costs	136			
easy-flow	140	255	339	
handling requirements	286			
properties	69			
mechanical properties	74			
melamine	43			
melt(ing)				
flow	84			
flow index	84	181	187	
index (MI)	12	13	84	151
	188			
rheometer	12			
test	154	185		
value	185			
point (MP)	5	61	62	83
temperature(s) (T _m)	5	6	23	24
	83	151	153	
metal inserts	237			
methyl pentenes				
(TPX resins)	98			

Index Terms

Links

milling				
thermoplastics	164			
thermosets	164			
modulus	71			
of elasticity	75			
secant	79			
shear	79			
moisture	196			
content	154	181		
molded-in				
color	173			
symbols	174			
mold(ing)	310			
“A” half	11			
“A” plates	315			
“B” half	11	20		
“B” plates	315			
process(es)	297	315		
process costs	142			
release	338			
release agents	155	287		
temperature	334	338	339	341
molecular weight	13			
distribution				
(MWD)	46	47		
monomer(s)	37	47		
cements	160			
molecule	40			

Index Terms

Links

N

neoprene	45	104		
nitrate	95			
nitrile	45			
rubber	104			
nonfill	340			
notch sensitivity	73			
nozzle	3	18	34	
assembly	7			
heater zone	8			
tip	7	8		
nylon	44	45	54	98
	154	250	257	
elastomers	117			
type (6)	6	10	115	
type (6/6)	6	10	229	
type (12)	116			

O

overhead costs	
fixed	141
variable	141
oxygen index	85

P

packaging	147	
requirements	275	287

Index Terms

Links

pad printing	172			
painting	167	168	287	
part				
cost	23			
“nesting,”	288			
quality	22			
volumes	298			
parting agents (<i>see mold release agents</i>)	155			
peripheral marking	170			
permeability	13			
phenolic	43	45	168	257
photopolymer	310			
pin-hinge	243			
plasma				
etch	167			
process	167			
plasticity	72			
plasticization	92			
plasticizers	54	55	155	
plating (electroplating)	168	287	238	
plunger	2			
device	33			
injection	54			
-type machine	3			
Poisson’s Ratio	79			
polyalkylene ether glycol(s)	105			
polyallomer(s)	6	10	99	
polyamide (nylon)	86	88		
polyamide-imide	6	10	45	

This page has been reformatted by Knovel to provide easier navigation.

Index Terms

Links

polyarylate	6	10	45	86
	88	102	118	250
polybutadiene	105			
polybutylene -terephthalate	6	10		
	106	119		
polycarbonate(s)	2	6	10	45
	86	88	96	99
	106	121	140	154
	156	167	168	229
	241	244	250	257
polychloroprene	104			
polychlorotrifluoroethylene (FCTFE)	97			
poly-dimethyl siloxanes	105			
polyester (PBT)	45	96	102	229
polyester (TP)	86	88	250	257
polyetheretherketone (PEEK)	6	10	45	99
polyetherimide	6	10	45	
polyethersulfone	45			
polyethylene	6	10	40	41
	45	55	56	99
	106	156	160	229
chlorosulfonated	104			
high-density (HDPE)	122	257		
low-density (LDPE)	252	257		
terephthalate (PET)	120			
polyethylene terephthalate (PET)	45	102		
polyfluoroethylene-propylene (PFEP)	97			
polyimide(s)	100			
polyketone(s)	123			

This page has been reformatted by Knovel to provide easier navigation.

Index Terms

Links

polymer(s)	37	40	44	46
	47	49		
amorphous	60			
crystalline	59	97		
liquid crystal	6	10	45	86
	88	98	114	250
structure	153			
sulfone	126			
polymerization	37	41	45	
	45			
	46			
	45			
polymethylpentene	6	10		
polyolefin(s)	99	166	287	
polyphenylene oxide (PPO)	6	10	45	86
	88	100	106	127
	229	250	257	
	6	10	45	86
sulfide (PPS)	88	100	128	250
	6	10	45	86
polypropylene	88	98	106	124
	156	160	229	250
	257			
	2	6	10	45
polystyrene	54	101	106	156
	257			
	125			
high-impact	125			
polysulfide(s)	45			

This page has been reformatted by Knovel to provide easier navigation.

Index Terms

Links

polysulfone(s)	6	10	86	101
	154	229	250	
rubber	104			
polytetrafluoroethylene (TFE)	97	98		
polyurethane(s)	45	106	168	
polyvinylchloride (PVC)	6	10	45	54
	55	96	101	129
	156	163	168	229
	257	336		
polyvinylidene fluoride	97			
pressure	4			
back	23	24		
clamp(ing)	11	13	14	339
closing	17			
ejection	338			
hold(ing)	11	13	18	
injection	11	23	24	26
	30	339	342	
process				
controls	22			
design	216			
implementation	216			
maintenance	216			
management	213	214		
product				
cost breakdown	208			
design	203	204	255	
checklist	279	280		
development	209			

This page has been reformatted by Knovel to provide easier navigation.

Index Terms

Links

product (<i>Cont.</i>)		
time line	216	217
requirements	223	227
project management	213	214
propionate	96	97
proportional limit	75	
prototype(ing)	307	
rapid (RP)	310	
virtual	310	

Q

quality	
costs	223
part	22
programs	298
rating	296

R

radii	278			
reaction injection molding (RIM)	106			
recycling	251			
symbol	251			
regrind	2	136	138	156
	198	334	336	338
reinforcement(s)	49	156		
fibers	52			
materials	51			
metal fiber(s)	52	53		

Index Terms

Links

resin grade	156			
resistivity				
surface	86			
volume	86	249	250	
rivets	241			
robotic systems	149			
roll-on decorating	169			
rubber(s)				
isoprene	103			
natural	103			
neoprene	104			
styrene-butadiene	1			
runner(s)	140	317	325	337
	341			
cross-section	321	322		
sizing	336			
system(s)	136	137	139	318
versus shot size	138			

S

safety	251			
screen printing	172	173		
screw				
concept	2			
injection machine	2	3	54	
retention	237			
self-tapping	237			
secondary operation(s)	146	149	150	286

<u>Index Terms</u>	<u>Links</u>			
Selective Laser Sintering (SLS)	311			
semicrystalline materials	246			
costs	146			
setup				
charges	144			
sheet	24	25		
sharp corners	278			
shear				
direct s	80	81		
strain	80			
stress	80			
shipment schedules	275			
short shots	340			
shot size	139			
shrinkage	26	28	30	65
	229	255	257	279
	341			
amorphous	29			
anisotropic	28			
control	29	32		
crystalline	29			
factors	229			
high	28			
impact	50	66		
isotropic	28			
low	28			
medium	28			
post-mold	30	31		

Index Terms

Links

shrinkage (*Cont.*)

rate	26	27	29	69
	70	73	74	181

silicone(s)	45	105		
-------------	----	-----	--	--

sink marks	255	261	341	342
------------	-----	-----	-----	-----

snap-fit	243	251		
----------	-----	-----	--	--

Society of Manufacturing Engineers

(SME)	331			
-------	-----	--	--	--

Society of Plastics Engineers (SPE)	2	289		
-------------------------------------	---	-----	--	--

Society of the Plastics Industry (SPI)	2	251	265	
--	---	-----	-----	--

Solid Ground Curing (SGC)	312			
---------------------------	-----	--	--	--

specific gravity	50	51	67	69
	181			

spin welding	159			
--------------	-----	--	--	--

splay	342			
-------	-----	--	--	--

sprue

bushing	317			
---------	-----	--	--	--

volume	138			
--------	-----	--	--	--

staking	246			
---------	-----	--	--	--

star diagrams	106			
---------------	-----	--	--	--

Statistical Process Control (SPC)	298			
-----------------------------------	-----	--	--	--

Stereolithography Apparatus (SLA)	310			
-----------------------------------	-----	--	--	--

stiffness	13	151	153	156
-----------	----	-----	-----	-----

strain	74	78		
--------	----	----	--	--

normal	75			
--------	----	--	--	--

shear	79			
-------	----	--	--	--

strength

compressive	188			
-------------	-----	--	--	--

and modulus	81			
-------------	----	--	--	--

This page has been reformatted by Knovel to provide easier navigation.

Index Terms

Links

strength (*Cont.*)

dielectric	87	180		
flexural	50	82	187	189
<i>s</i> and modulus	81			
impact	83	97		
IZOD	189			
tensile	13	50	51	66
	67	188		
ultimate	79			
yield	13	79		
stress(es)	74	76	78	88
	195	196	259	262
	337			
bending	195			
compression	195	259		
-crack resistance	13			
cracking	92			
direct	74			
molded-in	32	33		
normal	75			
shear	74	79	195	259
/strain	71	77		
tensile	259			
tension	195			
twisting	195			
structural foams	106	107		
styrene	41	229	309	
acrylonitrile (SAN)	6	10	130	
butadiene	6	10	45	

This page has been reformatted by Knovel to provide easier navigation.

Index Terms

Links

styrene (*Cont.*)

-maleic-anhydride (SMA)	131
subcontracting	298
surface finished	166

T

tapping

thermoplastics	160			
thermosets	164			
temperature(s)	4	46	47	88
barrel	29	336	340	342
controller	9	24		
glass transition(T_g)	62	83	192	197
heat deflection (HDT)	50	84	191	192
index	85			
maximum use	67			
melt(ing [T_m])	5	6	23	24
	153	192	197	
mold	8	23	24	334
	338	339	341	
process t requirements	61	63		
Vicat softening	13	193		
zones	5			
tensile modulus	67			
testing	179			
arc resistance	181	182		
ASTM #D-495	181			
compressive strength	187			

Index Terms

Links

testing (*Cont.*)

ASTM #D-695	187	
creep	189	
ASTM #D-618	180	
ASTM #D-674	189	
electrical	179	
density	183	
ASTM #D-792	183	
ISO #1183:1987	183	
dielectric constant	180	181
ASTM #D-150	180	
ISO #IEC	250	180
dielectric strength	180	
ASTM #D-149	180	
ISO #IEC	243	180
flammability	193	194
ASTM #UL-94	193	
ISO #UL-94	193	
flexural strength	188	
ASTM #D-790	188	
ISO #178:1993	188	
impact	189	
ASTM #D-256	189	
ISO #179 and 180:1993	189	
limiting oxygen index (LOI)	194	
ASTM #D-2863	194	
ISO #589:1984	194	
melt flow index	185	
ASTM #D-1238	185	255

Index Terms

Links

testing (*Cont.*)

ISO # 1133:1991	185	
melting point	189	
ASTM #D-3418	190	
ISO #3146:1985	190	
moisture content (Tomasetti Volatile Indicator [TVI])	185	
requirements	288	
shrinkage rate	182	
ASTM #D-955	182	
ISO #294-4	182	
surface resistivity	180	181
ASTM #D-257	180	
ISO # IEC 93	180	
tensile strength	187	
ASTM #D-638	187	
ISO #527-1 and 2:1993	187	
Vicat softening temperature	193	
ASTM #D-1525	193	
ISO #306:1987	193	
volume resistivity	180	181
ASTM #D-257	180	
water absorption	184	
ASTM #D-570	184	
ISO #62:1980	184	

test(s)

fatigue endurance	83	
impact	83	191
Charpy	83	189

Index Terms

Links

test(s) (*Cont.*)

falling dart	83			
IZOD	50	83	189	
tensile	83			
melt index (ASTM #D-1238)				
stress/strain (tensile)	77			
tetrafluoroethylene	6	10		
Texas Plastic Technologies	332			
textured surface	174			
texture(s)	175	230	231	277
	300	309		
thermal				
coefficient of expansion	50			
conductivity	85			
properties	83			
thermoforming	309			
thermogravimetric analysis (TGA)	200	201		
thermoplastic polyester				
(PBT)	6	10		
(PET)	6	10		
thermoplastic(s)	42	43	47	55
	163	230	248	255
copolyester(s)	102			
polyester(s)	102			
polyolefin	132			
polyurethane	133			
rigid	159			
rubbers	103			
thermosets	42	43	47	55

Index Terms

Links

time	4	46
clamp	15	
cooling	15	
initial injection	14	
injection	14	
injection hold	14	
mold-closed	15	
tolerance levels	297	
tool(ing)	326	
approval signoff	274	
build <i>t</i>	216	
charges	136	145
costs	144	
amortizing	145	
debug <i>t</i>	216	
design <i>t</i>	216	
build and repair	298	
prototype	285	
torque	236	
toughness	13	
transparency	70	
troubleshooting	331	343
tips	342	
turning		
thermoplastics	164	
thermosets	164	
twisting	33	
two-color (two-shot) molding	174	175

Index Terms

Links

U

ultrasonic

assembly machine	246		
energy	152	153	245
welding	151	245	287

ultraviolet light

absorbers	56
-----------	----

undercuts

Underwriters Laboratories (UL)	56	85		193
	194	249	273	277
	288	289		

United Silicone, Inc.

urea

urethane(s)

elastomer	6
-----------	---

U.S. Food and Drug

Administration	
(FDA)	277

V

vacuum metallizing

process	169
---------	-----

vendor(s)

dedicated specialty	293	
full-service	293	294
qualification form	300	

venting

341

Index Terms

Links

vertical

molding

20

21

stamping

170

vibration welding

152

Vicat softening point

84

vinyl

252

virgin

138

336

material

2

334

viscosity

66

W

wall thickness

255

262

263

278

warpage

33

66

67

262

342

water

absorption

70

181

jet cutting

166

weathering

92

work-in-process (WIP)

147

Y

yield point

75